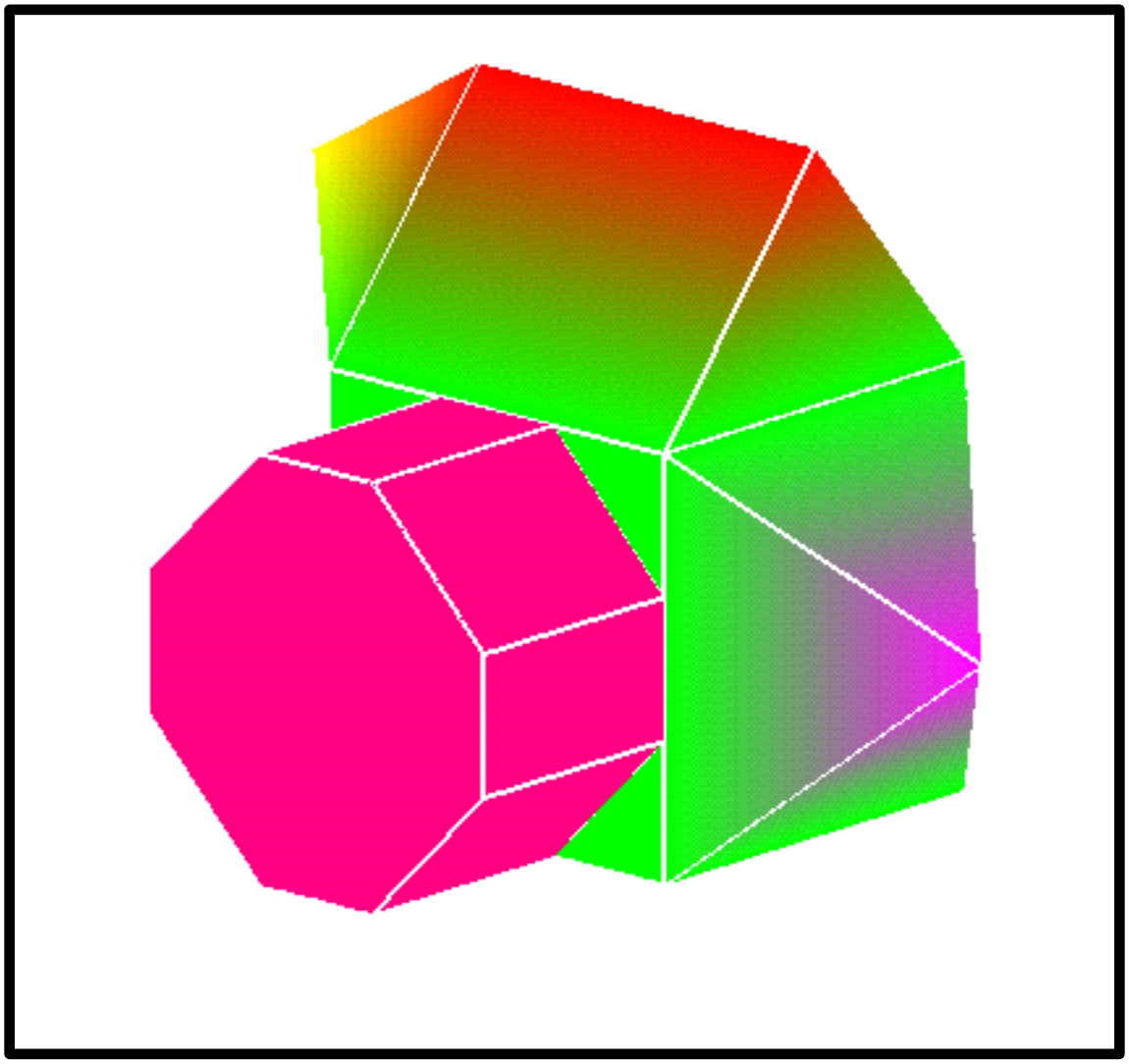
LAUR 95−2986

Manual

GMV *Version 2.9*

General Mesh Viewer

User's Manual



By Frank A. Ortega

Los Alamos

NATIONAL LABORATORY

Los Alamos National Laboratory is operated by the University of California for the United States Department of Energy under contract W−7405−ENG−36

Edited by Patricia W. Mendius, Group CIC−1

An Affirmative Action/Equal Opportunity Employer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither The Regents of the University of California, the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by The Regents of the University of California, the United States Government, or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of The Regents of the University of California, the United States Government, or any agency thereof.

Table of Contents

1.0 Getting to Know GMV ***1−1***

1.1 Starting GMV **1−1**

1.2 The GMV Temporary Field Files **1−1**

1.3 The GMV User Interface **1−2**

The mouse controls **1−2**

Using menus **1−2**

Buttons **1−3**

Slider Bars **1−5**

Scroll bars **1−5**

1.4 The Main GMV Window **1−5**

The menu bar **1−5**

Twist, elevation, and azimuth **1−5**

Axes orientation viewbox **1−5**

Magnification slider **1−6**

1.5 The File Selection Menu **1−6**

1.6 The GMV Resource file, gmvrc **1−8**

gmvrc keywords **1−8**

Sample gmvrc file **1−9**

2.0 The File Menu ***2−1***

2.1 Read GMV File **2−1**

New Simulation **2−1**

Same Simulation **2−1**

Same Simulation, Same Cells **2−1**

Auto Read − Same Simulation **2−2**

Auto Read − Same Simulation, Same Cells **2−3**

2.2 Put and Get Attributes **2−4**

2.3 Save gmvrc **2−4**

2.4 Snapshot **2−4**

2.5 Quit **2−5**

3.0 The Display Menu ***3−1***

3.1 Nodes **3−1**

Viewing nodes, their vectors, and numbers **3−1**

Vectors **3−2**

Selecting nodes to display **3−3**

Selecting nodes by Materials and Flags **3−3**

Selecting nodes by Node Field Data Range **3−4**

Selecting nodes by Search Sphere 3−5
Selecting nodes by Number(s) 3−5
Selecting nodes by Search Box 3−6
Selecting nodes by Groups 3−6
Activating node selection 3−6
3.2 Cells 3−7
Viewing cell faces, edges, and numbers 3−7
Coloring cells by materials, fields, and flags 3−8
Cell vectors 3−8
Face vectors 3−8
Selecting cells to display 3−8
Explode 3−9
3.3 Polygons 3−10
Shading and outlining polygons 3−10
Selecting materials to display 3−10
Changing explode percentage 3−11
Selecting a polygon subset 3−11
Changing material order 3−11
3.4 Tracers 3−12
Methods of displaying tracers 3−12
Selecting data field for tracer to represent 3−13
Selecting tracers to display 3−13
Display tracer history 3−14
3.5 Surfaces 3−14
Viewing surface faces, edges, and numbers 3−15
Coloring surfaces by materials, fields, and flags 3−15
Surface vectors 3−16
Selecting surfaces to display 3−16
Explode 3−16

4.0 The Calculate Menu ***4−1***
4.1 Average 4−1
Selecting weighting field 4−1
Selecting a field to Average 4−1
4.1 Cutlines 4−1
Selecting a cutline 4−2
Creating a cutline 4−2
Cutline display options 4−3
Cutline 2D plot 4−3
4.2 Cutplanes 4−3
Main Cutplanes Menu 4−4
Value 4−4

Node or Cell Field 4−4
New Field 4−5
Apply Field Change 4−5
Cutplane Selection Buttons 4−5
Cutplane Description Menu 4−6
Clip on Field Subset and Cell Selection 4−6
Cutplane Options 4−6
Faces 4−6
Coutour Lines 4−7
Edges 4−7
Height 4−7
Distance 4−7
Adding a cutplane to the main viewer 4−8
Adding a cutplane the easy way 4−8
4.3 Cutspheres 4−9
4.4 Distance 4−9
4.5 Field Calc 4−9
Selecting a field to build 4−9
Build (calculate) the new field 4−9
4.6 Grid Analysis 4−10
Selecting cells by nodes or cell numbers 4−11
Color By: 4−12
Median and Voronoi mesh 4−12
4.7 Isosurfaces 4−12
Adding a material isosurface 4−12
Adding a field isosurface 4−13
Clip on field subset and cell selection 4−15
Coloring isosurfaces with field values 4−15
4.8 Isovolume 4−15
4.9 Query Data 4−16
Getting node and cell values 4−16
Probing node and cell numbers 4−16
Getting node numbers by field value 4−17

5.0 The Ctl−1 (Controls−1) Menu ***5−1***
5.1 Animation (ortho. and perspective modes) 5−1
Number of animation frames 5−1
Rotation 5−2
Center translation 5−2
Magnification 5−2
Vector flow 5−2
Cutplane 5−3

Fade 5−3
Explode during animation 5−4
Snapshot 5−5
Quick look 5−5
Isosurface animation 5−6
Cutsphere animation 5−7
5.2 Animation (flight mode) 5−8
Setting control points 5−8
Saving and Retrieving control points 5−9
Quick look and snapshot 5−9
5.3 Axes 5−9
5.4 Beep Sound 5−9
5.5 Bounding Box 5−9
5.6 Center 5−10
Center on node or cell 5−11
5.7 Clip 5−11
5.8 Color Bar 5−12
Turning on 5−12
Preferences 5−12
5.9 Color Edit 5−12
Materials, Isosurfaces, Isovolume 5−13
Changing material or isosurface colors 5−14
Reinstating default colors 5−14
Field data colormap 5−14
Background Color 5−15
5.10 Contour Levels 5−15

6.0 The CTL−2 (Controls−2) Menu ***6−1***
6.1 Cycle 6−1
6.2 Data Limits 6−1
Fields 6−1
Tracers 6−2
6.3 Distance Scale 6−2
6.4 Interactivity 6−3
6.5 Light 6−3
6.6 Line Width 6−3
6.7 Point Size 6−4
6.8 Plot Box 6−4
6.9 Scale Axes 6−5

7.0 The CTL−3 (Controls−3) Menu ***7−1***
7.1 Subset 7−1

Nodes, cells, tracers 7−1
Polygons 7−1
7.2 Texture smoothing 7−1
7.3 Time 7−2
7.4 Title 7−2
7.5 Use Display List 7−2
7.6 Vector Control 7−3
7.7 Virtual Trackball 7−3
7.8 Window Size 7−3
7.9 Zoom (Rubberband) 7−4

8.0 The Reflections Menu ***8−1***
8.1 Reflecting about an axis 8−1

9.0 The View Menu ***9−1***
9.1 Orthographic 9−1
9.2 Perspective 9−1
9.3 Flight 9−1
9.4 Stereo Perspective 9−1
9.5 Stereo Flight 9−2

10.0 The GMV Input Format ***10−1***
10.1 Input Specifications 10−1
10.2 Input Data Details 10−9
Header 10−9
Nodes 10−10
Nodev 10−10
Cells 10−10
Vaces 10−11
Faces 10−11
Nodeids 10−12
Cellids 10−12
Faceids 10−12
Materials 10−12
Velocities 10−12
Variables 10−13
Subvars 10−13
Flags 10−13
Polygons 10−14
Tracers 10−14
Tracerid 10−14
Problem Time 10−14

Cycle Number 10−14
Surfaces 10−14
Surface Materials 10−15
Surface Velocities 10−15
Surface Variables 10−15
Surface Flags 10−15
Surfids 10−15
Element groups 10−15
Comments 10−16
Codename, Codever, Simdate 10−16
10.3 Reading some GMV data from a different file 10−16
10.4 Sample Input Data 10−17

11.0 Making Movies With GMV ***11−1***
11.1 GMV Command Line Options 11−1
11.2 GMV Movie Utility 11−3

12.0 GMVBATCH ***12−1***
12.1 GMVBATCH − Off screen rendering 12−1
12.2 GMVBATCH Command Line Options 12−1

13.0 Helpfull Hints ***13−1***

Illustrations Listing

1−1 Mouse button schematic
1−2 Main GMV window
1−3 Main GMV window continued
1−4 Twist, Elevation, & Azimuth controls
1−5 Axes view box
1−6 Magnification & Interactivity controls
1−7 The File Selection menu
2−1 Auto Read menu
2−2 Auto Snapshots menu
2−3 SnapShot menu
3−1 Nodes menu
3−2 Node Field Selection menu
3−3 Build Vector submenu
3−4 Node Select submenu
3−5 Node Materials and Flags submenu
3−6 Node Field Data Range submenu
3−7 Node Search Sphere submenu
3−8 Node Number submenu
3−9 Node Search Box submenu
3−10 Cells menu
3−11 Cell Color By submenu
3−12 Cell Materials and Flags submenu
3−13 Cell Explode submenu
3−14 Polygons menu
3−15 Polygon Subset submenu
3−16 Material Order submenu
3−17 Tracers menu
3−18 Tracer Select menu
3−19 Surface menu
4−1 Average menu
4−2 Cutline Selection menu
4−3 Create cutline menu
4−4 Main Cutplanes menu
4−5 Cutplane menu
4−6 Cutplane Options menu
4−7 Field Calc. Selection menu
4−8 Field Calc. Build menu
4−9 Grid Analysis menu
4−10 Material Isosurface menu
4−11 Isosurface demo
4−12 Field Isosurface menu
4−13 Isovolume menu
4−14 Query Data menu
4−15 Get Node by Field Value menu
5−1 Animation menu
5−2 Cutplane Animation submenu
5−3 Fade Animation submenu
5−4 Explode Animation submenu
5−5 Isosurface Animation submenu
5−6 Cutsphere Animation submenu
5−7 Flight Animation menu
5−8 Bounding Box menu
5−9 Center menu
5−10 Center on node menu
5−11 Clip slider controls
5−12 Color Bar
5−13 Materials, Isosurfaces, Isovolume color edit menu
5−14 Field Colomap Selection menu
5−15 Background Color menu
5−16 Contour Levels menu
6−1 Data Limits menu
6−2 Interactivity menu
6−3 Light Control menu
6−4 Line Width menu
6−5 Point Size menu
6−6 Plot Box menu
6−7 Scale Axes menu
7−1 Subset menu
7−2 Title menu
7−3 Vector Control menu
7−4 Window Size menu
7−5 Zoom (Rubberband) menu
10−1 Cell vertex order
11−1 GMV Movie menu

Preface

Revision Record for GMV:

Revision/Date:	Description:
January 2002	Version 2.9

Description and Intent:

This manual describes the General Mesh Viewer (GMV).

GMV is a three−dimensional visualization tool that can process data from any 3−D mesh. Data to be visualized are taken from a properly formatted input file and displayed on the screen. With simple pull down menus, windows, and mouse controls, many special functions are available to maximize the practical value of any simulation GMV may be asked to visualize.

This manual has three purposes: to teach the beginner how to operate in the GMV environment, to act as a reference guide for the more experienced user, and to define the format for the GMV input file.

The only possible prerequisite for the use of GMV is the knowledge of a computer programming language so that you can write code to generate GMV input data. However, finished input files are not very difficult to obtain and can also be written manually using a text editor.

Syntax Conventions:

Words enclosed in double quotes ("like this"), unless otherwise stated, are actual quoted material from the GMV environment, such as menu options or error messages. This punctuation does not apply to the description of the GMV input format. The GMV input file section has its own set of conventions.

Getting to Know GMV

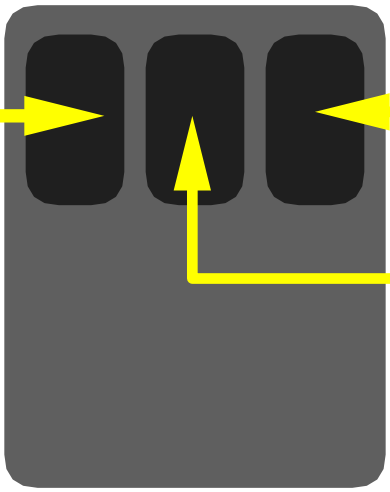


Figure 1−1. Mouse button schematic

Starting GMV:

To start GMV, the location of the executable must be known. If GMV is in the current path, simply type:

`gmv`

on the console and press the enter key. The main GMV window will appear first, followed by a File Selection menu requesting the name of a GMV input file. Double click on the name of an input file, or select the file and click on "OK." If the requested file is not a valid GMV input file, then a box stating this fact will appear and allow you another chance to select a valid GMV input file. After the file is chosen, the mouse pointer will change into a watch, indicating that GMV is processing the input file and preparing to display the data on the screen. Be patient, this may take a few seconds or even minutes depending on the size of the input file. An object will then be displayed on the screen. The object displayed depends on the input data and the following order: polygons, cells, nodes. If polygons exist, they are drawn first. If no polygons exist, then cells are drawn. If no cells exist, then nodes are drawn, which must exist in a valid input file. In addition, the "Display" window corresponding to whatever was displayed first will pop up. For example, if the input file contains polygon data, the "Polygons" window will automatically pop up when the input file is first opened.

The GMV temporary field files:

Upon the opening of any input file, GMV creates temporary files on the local system which hold node field data, cell field data, and polygon data . These files are not visible to the user. GMV will first attempt to place these files in the

directory specified in the environment variable "TMPDIR." Set this directory with the C shell command:

setenv TMPDIR *directory_name*

where *directory_name* is the path where you want GMV to place the field data file. If "TMPDIR" is undefined, GMV will attempt to put the field data file in "/usr/tmp." These temporary files are removed when GMV ends.

The GMV user interface:

The mouse controls

The mouse controls for GMV have been designed for maximum ease of use with a three button mouse, (see Fig. 1−1). The left mouse button has two different functions, depending on whether the cursor is in a menu area or the viewing area. In the case when the mouse pointer is in the menu area, the left mouse button is used to pull down menus, select options, drag slider bars, etc. When the mouse pointer is in the viewing area, the left mouse button functions as a rotation device. For example, (assuming twist is set to zero) while holding down the left mouse button and dragging left or right, the object in the viewing area rotates either left or right, depending on the current orientation of the axes. Moving the mouse up and down in this manner rotates the object either up or down, again depending on the current placement of the axes. The middle mouse button provides a panning function. Holding the middle button and moving the mouse shifts the object linearly in any direction without any rotation. For example, while holding down the middle mouse button and dragging right, the object moves to the right. Finally, the right mouse button is used for controlling the magnification of the object in the main viewer. For example, while holding down the right mouse button and dragging up, the object grows larger. Dragging down causes the object to appear smaller. Motion to the left or right does nothing.

Using menus

The top row of the GMV window is lined with various menus. To open a menu, click the left mouse button on the name of the menu desired. A small box will appear with menu options. To select a menu option, again click the mouse on the desired option. Some of the menu options will open sub menus for specific program functions which require additional information. To choose from a submenu, click on the original option and move the mouse to the right and follow the usual rules for choosing from menus.

Buttons

GMV has three different types of buttons which are used to select various functions: regular buttons, toggle buttons, and radio buttons. Regular buttons are fairly large and have labels inside such as "CLOSE" or "CANCEL." To activate these, just click the mouse on the button desired. It will temporarily depress to indicate that it has been activated. The toggle buttons used in GMV are small and square in shape with labels next to them. These buttons have two different states, on or off. The little indented square will appear yellow when it is on, and grey when it is off. Radio buttons are a set of toggle buttons that allow only one selection of the set and are diamond shaped.

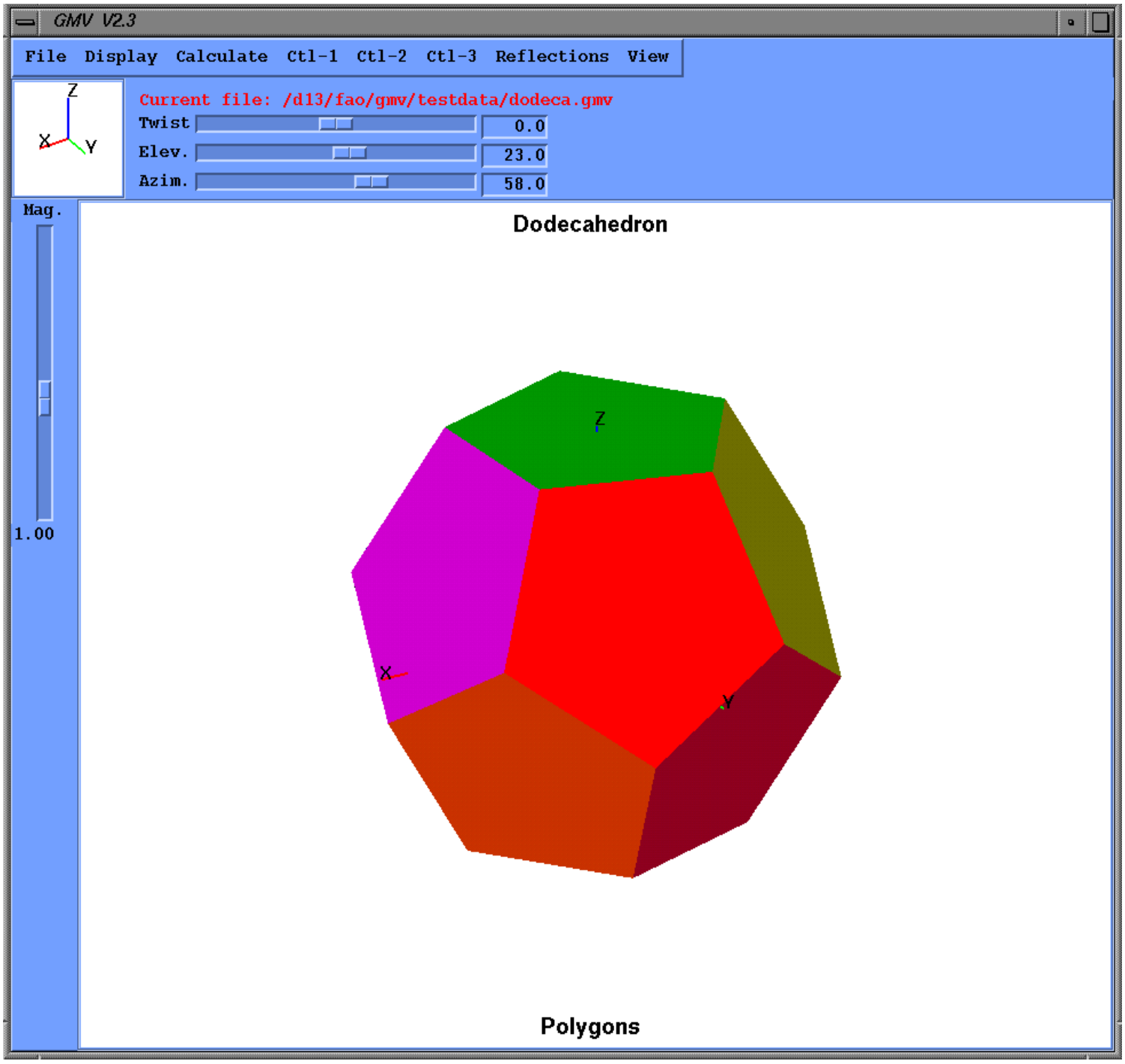
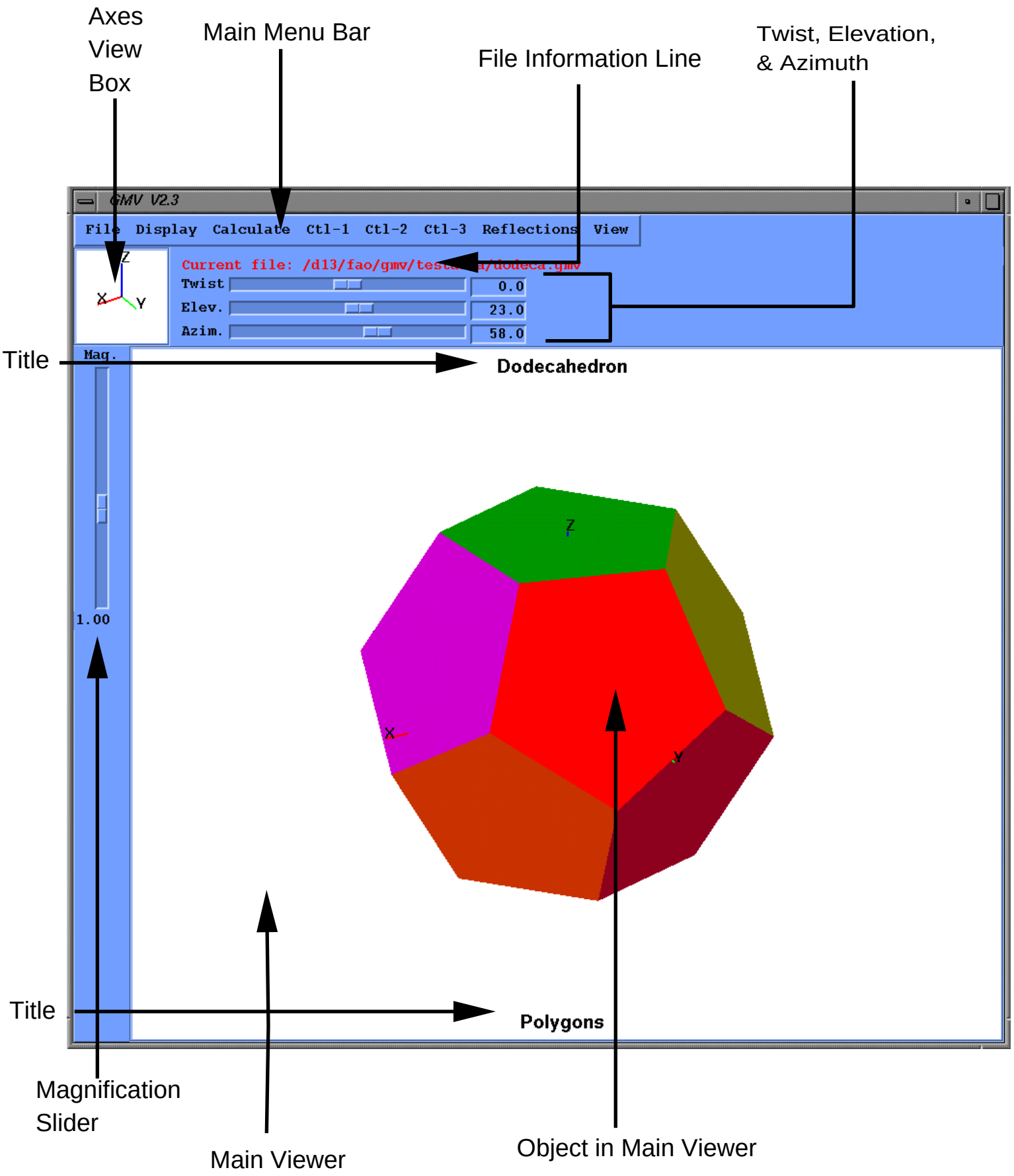


Figure 1−2. Main GMV window


Axes View Box
Main Menu Bar
File Information Line
Twist, Elevation, & Azimuth
GMV V2.3
File Display Calculate Ctl-1 Ctl-2 Ctl-3 Reflections View
Current file: /d13/fao/gmv/tes
Twist 0.0
Elev. 23.0
Azim. 58.0
Mag.
1.00
Title
Dodecahedron
Z
X
Y
Title
Polygons
Magnification Slider
Main Viewer
Object in Main Viewer

Slider bars

Slider bars are control devices used throughout the GMV user interface. The use of slider bars is very easy. Just click and hold the left mouse button on the rectangular shaped slider control and drag it back and forth until the desired adjustment has been made. When more precise changes are desired, you can click the left mouse button on a portion of the slider bar not covered by the slider control and the slider will move at predetermined units.

Scroll bars

Many menus within the GMV environment contain lists of things to choose from. Lists are placed into scroll boxes. On the right side of a scroll box is the scroll bar. The scroll bar is much like a slider bar. To scroll through a list, click and drag the slider back and forth in its track until the desired part of the list is in view. You may also click in the scroll bar's track on either side to move through the list more slowly.

The main GMV window: (see Fig. 1−2 & 1−3)

The menu bar

The menu bar is located at the very top of the main GMV window. The menu names listed in order are: file, display, calculate, ctl−1 (controls−1), ctl−2 (controls−2), ctl−3 (controls−3), reflections, and view.

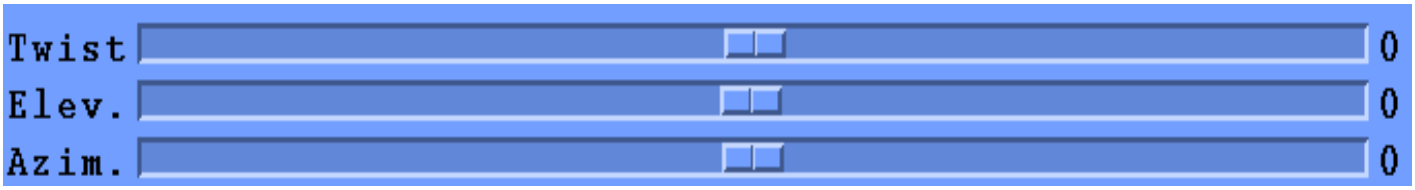


Figure 1−4. Twist, elevation, & azimuth controls

Twist, elevation, and azimuth

These three slider bars are located above the main viewer and control the viewing angle, (see Fig. 1−4). "Azimuth" is the angle on the X−Y plane measured from the X−axis. It has the same effect as using the left mouse button and moving left and right. "Elevation" is the angle in the direction of the Z−axis measured from the X−Y plane. It has the same effect as using the left mouse button and moving up and down. The "Twist" adjustment cannot be done with the mouse. The twist slider rotates the object about the X−axis.

Axes orientation view box

This box is located up and to the left of the light source box, (see Fig. 1−5). The box shows the orientation of the X, Y, and Z axes at all times, even if the

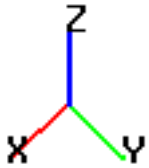


Figure 1−5. Axes view box

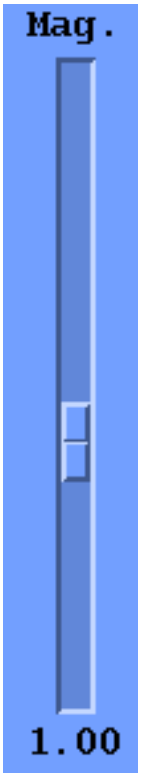


Figure 1−6. Magnification control

axes in the main viewer are turned off. It is used mainly for reference.

Magnification slider bar

The magnification slider bar (Fig. 1−6) is labeled "Mag" on the top and has the magnification factor displayed at the bottom (1.00 is default). Sliding the bar up and down changes the size of the object in the main viewer. The same effect can be accomplished with the right mouse button, (see **The mouse controls**).

The File Selection Menu:

GMV opens with a File Selection Menu (see Fig. 1−7). Use this menu to select a file to process. Near the top of the window is a box labeled "Filter." The filter controls what type of files will be displayed in the "Files" box below. For example, a filter such as "/usr/people/guest/*.inp" would display only files in the directory /usr/people/guest that have the extension ".inp". A filter is very useful for sorting GMV input files from the other files in that particular directory. Clicking on the "Filter" button near the bottom of the window activates the current filter. Pressing "Enter" on the keyboard while the mouse pointer is in the filter box has the same effect.

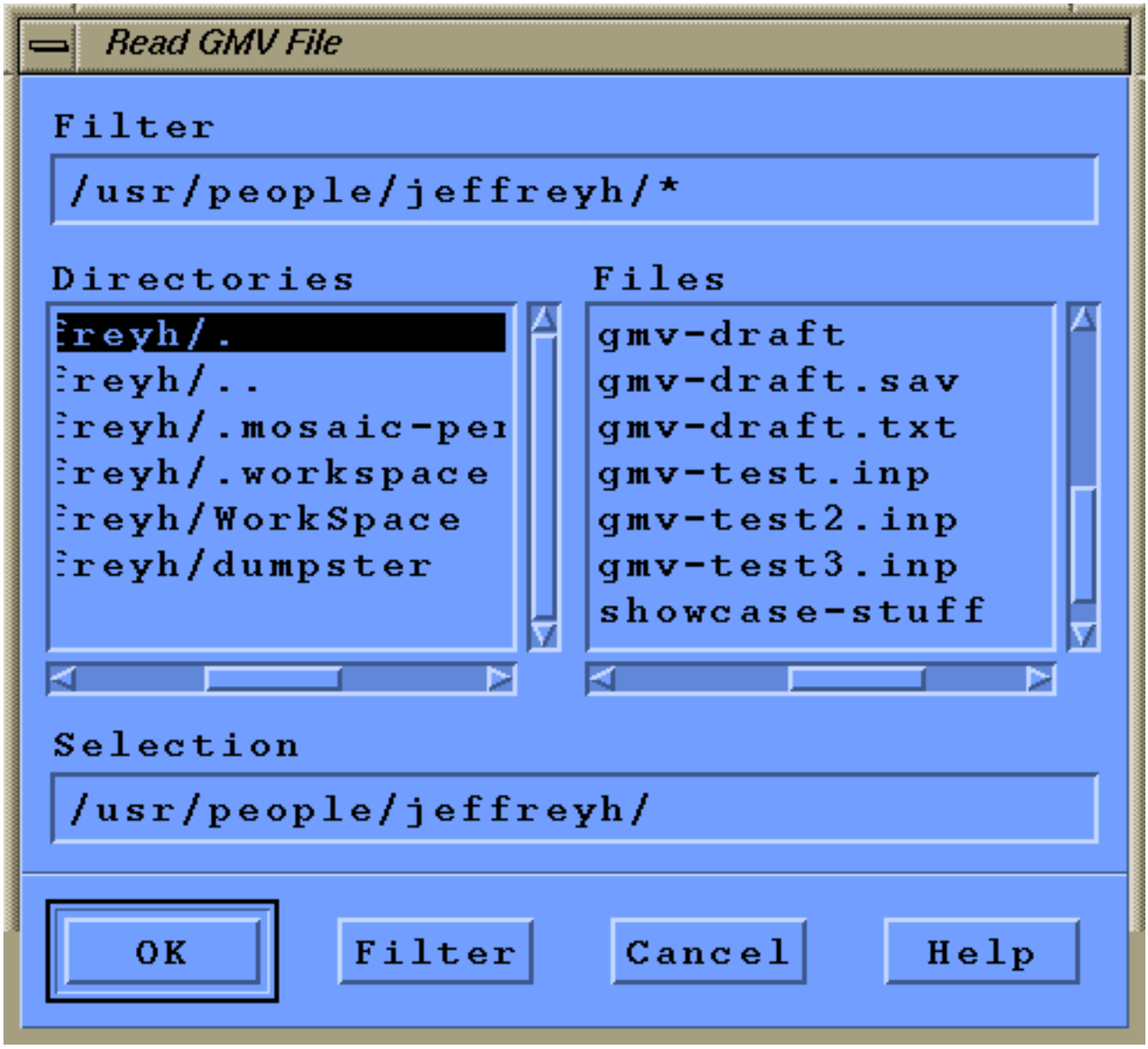


Figure 1−7. The File Selection menu

Once the directory with the GMV input file has been located, you must choose from the list of files in the "Files" box. To choose a file, use the scroll bar to position the file name within view and click on the file's name once. The name will appear highlighted in the "Files" box and will also be copied down to the selection box near the bottom. If the exact location of the file is known, the name can be typed into the selection box manually. Now click on OK to launch the file into GMV. The previous steps may be skipped if you simply double click on the file name.

If, after the file has been chosen, a watch appears, the selected file is a pointsize − followed by 2, 4, 6, or 8. Sets point size in pixels.valid GMV input file and GMV is processing it. If the file is not a valid input file, a message box will appear stating this, and another opportunity will be given to choose a file.

The GMV resource file, gmvrc:

The file gmvrc is a GMV resource file that contains a set of generic drawing instructions that affect the initial display of a simulation. When the first input file is read, or when a file for a new simulation is read, GMV will look for the gmvrc file in the directory where GMV is started. If gmvrc is not found in the current directory, then GMV will look for gmvrc in the user's home directory. If a gmvrc file is found, then the display command settings in gmvrc will override the display defaults. Note that if a gmvrc command asks to display an object that does not exist, then the display may be blank. For example, if gmvrc asks for polygons to be displayed and polygons do not exist, then the image will be blank.

gmvrc keywords

The drawing options available in gmvrc are all keyword driven. The keywords and their options are:

gmvrc − indicates the start of a gmvrc file, required.

azim − followed by a floating point number between −180 and 180. Sets the azimuth angle.

elev − followed by a floating point number between −180 and 180. Sets the elevation angle.

twist − followed by a floating point number between −180 and 180. Sets the twist angle.

mag − followed by a floating point number. Sets the magnification.

nodes − followed by on or off. Sets nodes display on/off.

nodenumbers − followed by on or off. Sets node numbers display on/off.

cellfaces − followed by on or off. Sets cell faces display on/off.

celledges − followed by on or off. Sets cell edges display on/off.

cellnumbers − followed by on or off. Sets cell numbers display on/off.

polygons − followed by on or off. Sets polygons display on/off.

polygonlines − followed by on or off. Sets polygon lines display on/off.

axis − followed by on or off. Sets axis on/off.

time − followed by on or off. Sets time on/off.

cycle − followed by on or off. Sets cycle on/off.

linesize − followed by 1, 2, or 3. Sets line width in pixels.

linetype − followed by regular or smooth. Sets line type.

pointsize − followed by 2, 4, 6, or 8. Sets point size in pixels.

pointshape − followed by square or round. Sets point shape.

ncontours − followed by a positive integer. Sets the number of contour levels.

xreflect − followed by on or off. Sets reflection about x axis on/off.

yreflect − followed by on or off. Sets reflection about y axis on/off.

zreflect − followed by on or off. Sets reflection about z axis on/off.
xscaleaxis − followed by a positive floating point number. Sets the scale factor in the x direction.
yscaleaxis − followed by a positive floating point number. Sets the scale factor in the y direction.
zscaleaxis − followed by a positive floating point number. Sets the scale factor in the z direction.
boundingbox − followed by on or off. Sets bounding box on/off.
boundingboxcoords − followed by on or off. Sets bounding box coordinates on/off.
background_red − followed by a positive floating point number between 0 and 1. Sets the red component of the background color.
background_green − followed by a positive floating point number between 0 and 1. Sets the green component of the background color.
background_blue − followed by a positive floating point number between 0 and 1. Sets the blue component of the background color.
display_list − followed by on or off. Sets display list on/off.
trackball − followed by on or off. Sets the virtual trackball on/off.
beepsound − followed by on or off. Sets the GMV beep sound on/off.
distscale− followed by on or off. Sets the distance scale on/off.
windowwidth − followed by a positive integer. Sets the window width.
windowheight − followed by a positive integer. Sets the window height.
textureflag− followed by on or off. Sets texture smoothing on/off.
attributes − followed by a gmv attributes file filename. Specifies an attributes file to read for initial image.
end_gmvrc − indicates the end of the gmvrc file, required.

These keywords all have functional interactive counterparts that are available via menus and should be recognizable. Note that gmvrc and end_gmvrc are the only required keywords, all others are optional. If GMV encounters the attributes keyword, and the file does not exist, then it is ignored. Fully qualified attributes filename must be specified if the attributes file is not in the current directory. Placement of the attributes keyword in gmvrc is important, attributes will override previous options, while subsequent gmvrc options will override attributes. The best way to create a gmvrc file is to have GMV generate one for you with the "Save gmvrc" option in the "Files" menu A gmvrc file will be generated with the options set to those used by the current image.

Sample gmvrc file

```
gmvrc
azim −120.000000   elev 20.500000   twist 0.000000
mag 1.000000
nodes off   nodenumbers off
```

```
cellfaces off   celledges off   cellnumbers off
polygons on   polygonlines off
axis off   time off   cycle off
linesize 1   linetype regular
pointsize 2   pointshape round
ncontours 10
xreflect off   yreflect off   zreflect off
xscaleaxis 1.000000   yscaleaxis 1.000000   zscaleaxis 1.000000
boundingbox on   boundingboxcoords off
background_red 1.000000   background_green 1.000000
background_blue 1.000000
end_gmvrc
```

The File Menu

Read GMV file:

The first option in the "File" menu is "Read GMV File." Choose this option, and a pull down menu will appear. The pull down menu contains three options: New Simulation, Same Simulation, and Auto Read − Same Simulation.

New Simulation

The New Simulation option enables GMV to read a file that was generated by a different simulation than the simulation that created the current GMV input file. Selecting this option will display the File Selection menu (see Fig. 1−10) to select the next input file. Then the current custom menus will be destroyed while new custom menus for the data on this input file will be created. The first image will display either nodes, cells, or polygons following the rules used when GMV is started (see Chapter 1). Only the view angles, magnification, and material colors will be the same as the last image from the previous GMV file. The 3−D plot box, the subset ranges, and the field data ranges will all be reset to reflect the data in the new GMV file.

Same Simulation

The Same Simulation option allows GMV to read a file that was generated as a different time step from the same simulation as the simulation that created the current GMV input file but with a different cell configuration. Selecting this option will display the File Selection menu (see Fig. 1−10) to select the next input file. The current custom menus are not destroyed . The image displayed after reading the input file will contain exactly the same attributes as the image from the previous file.

The new image will be displayed much faster after the file is read since the custom menus do not have to be recreated. Also, any cutlines, cutplanes, isosurfaces, and isovolumes that existed in the previous image will be automatically calculated and displayed. The 3−D plot box, the subset ranges, and the field data ranges remain the same for successive implementations of the Same Simulation read option unless manually reset, an attributes file is read, or until a file is read with the New Simulation read option.

Same Simulation, Same Cells

The Same Simulation, Same Cells option is similar to the Same Simulation option, except that the cell configuration must be the same as the current GMV file. In other words, the cells must contain the same node numbers. A new cell face list and cell edge list will not be recalculated.

The new image will be displayed much faster after the file is read since the custom menus do not have to be recreated and cell faces and edges remain the same.

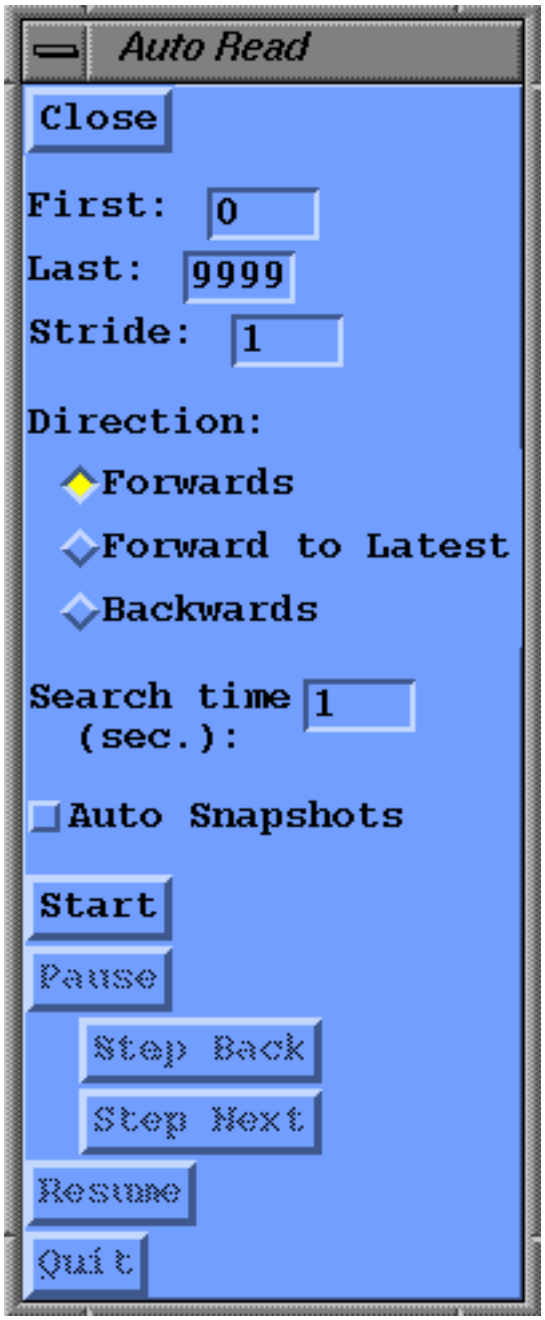


Figure 2−1. Auto Read Menu

Auto Read − Same Simulation

The Auto Read − Same Simulation option allows GMV to automatically read a time series family of input files created from the same simulation but with possibly different cell configurations. The filenames to be read by this option must all be the same except for a numeric suffix. The numeric suffix must all be either a 3 or 4 digit number within the family. Also the current GMV file must be a member of the family.

The files are read in a user determined sequence and the image produced after a file is read will have the same attributes as the last image from the previous file. As in the read Same Simulation option, it is important to have the plotbox, field data range, and subsets set to values that reflect data for the family of files. The attributes can be changed, however, by pausing the sequence and manually setting attributes or reading an attributes file. You can then resume the sequence or start over.

Selecting this option will display the Auto Read menu (see Fig. 2−1), unless the current file does not contain a numeric suffix. In the "First": and "Last:" text boxes, enter the range of input files to read. Enter a stride (skip value) in the "Stride:" text box. Next, select one of the direction options.

The "Forwards" direction option reads files form first to last incremented by the stride. The "Forward to Latest" option looks for the latest existing file within the specified range. This option is useful to view the latest complete GMV file as the files are being generated by a simulation code. The "Backwards" option reads files from last to first decremented by the stride.

In the "Search time (sec):" text box, enter the time interval GMV will use to search for the next file in the sequence, not including the time to read a file. To sweep through a series of files as fast as possible use a 1 second interval If you would like snapshots of the first image displayed after the next file is read, press the

"Auto Snapshots" button to display the control menu. See below for the description of the Auto Snapshot menu.

Press the "Start" button to start reading files. Press the "Pause" button to pause the file search. Use "Pause" when you want to closely inspect the current image or when you want to change the current image. While in "Pause", use the "Step Back" or "Step Next" button to view the previous or next image. Press the "Resume" button to continue the file search after a pause. The "Quit" button stops the file search. Note that if you change the image during the search, subsequent images will use the image attributes of the image you changed.

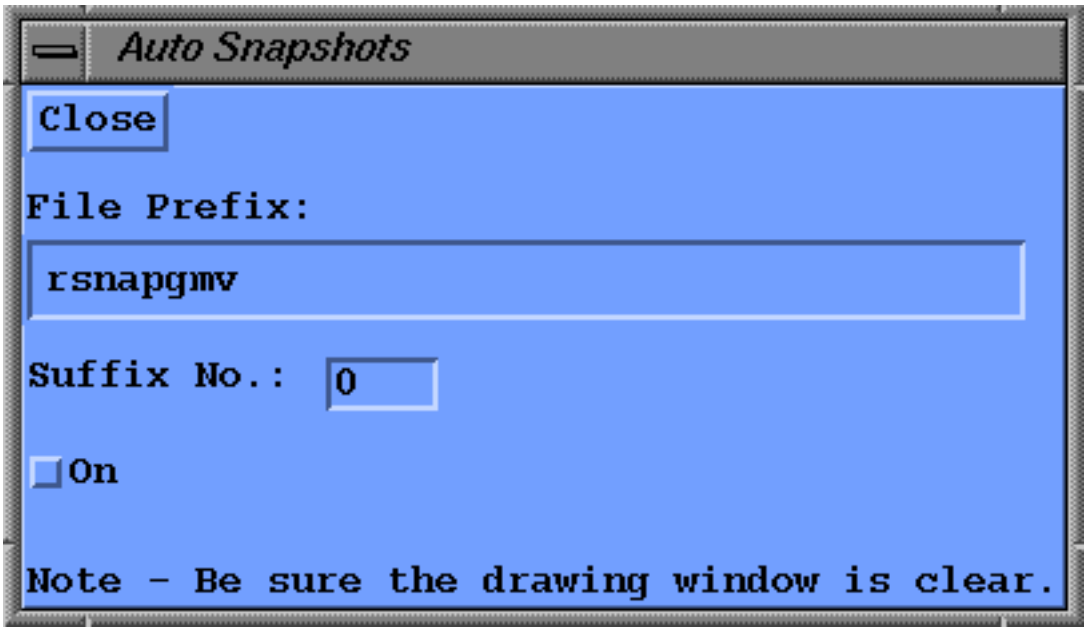


Figure 2−2. Auto Snapshots Menu

To automatically create a snapshot from each file, press the "Auto Snapshots" button and the Auto Snapshots menu appears (see Fig. 2.2). Enter the file prefix for the rgb files in the "File Prefix" text area. Enter the start of the 4 digit suffix number sequence in "Suffix No.:" and the "On" button to create snapshots when starting or resuming Auto Read. Be sure that the drawing window is not obstructed during Auto Snapshots.

Auto Read − Same Simulation, Same Cells

The Auto Read − Same Simulation, Same Cells option except that the cell configuration must be the same as the current GMV file. New cell face lists and cell edge lists will not be recalculated.

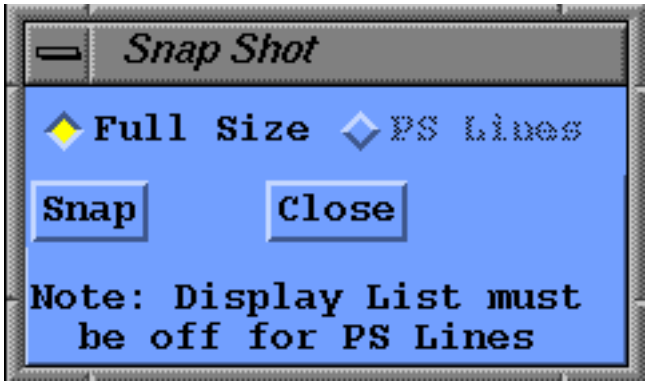


Figure 2−3. SnapShot menu

Put and get attributes:

Attributes are the collective sum of all the options available in GMV, Normally when GMV is first run, all of GMV's options are set to their defaults. For example, the "Twist," "Elevation," and "Azimuth" slider bars are all set to zero and the magnification factor is set to one. However, suppose you have worked on viewing the object from a certain angle and have created isosurfaces and a cutplane, or you want to apply the attributes to a different time step of the same simulation. If you want these to reappear the next time you start GMV, you can save the attributes in a file and retrieve them later. Choosing the "Put Attributes" option brings up a file selection menu. Enter a file name then click OK to save the file. The "Put Attributes" function is necessary in order to create time sequence movies of a simulation (See **The GMV Movie utility**). The attribute file may be retrieved later by invoking the "Get Attributes" option under the "File" menu. Take note that when a set of attributes is saved and then immediately retrieved, the name of the attribute file will not show up in the "Get Attributes" list of files until the "Filter" button is clicked, updating the file list.

Save gmvrc:

Save gmvrc will save a gmvrc file in the directory where GMV was started. The current generic display options will be saved in gmvrc.

Snapshot:

Snapshot is a tool that can create image files of the currently displayed GMV data, (see Fig. 2−3). A GMV snapshot can only be two sizes: full size and movie size. Movie size (720 x 486) is the smaller one and can be used to create a sequence of images for animation. There is also an option to create a PostScript file of only lines (Display List must be off for PS Lines). Snapshot images are always saved in the SGI−RGB format. There are many freeware utilities available to convert to other formats.

After the "Snapshot" option has been chosen, a small window will appear. "Full Size and "PS Lines" options appear. Choose one. The radio button

next to the currently selected snapshot size will be yellow. When all is ready, click on "Snap" and a File Selection Menu will appear. Enter a file name and click on the "OK" button to save the snapshot. After the image is stored, click on the "Close" button and continue work. The snapshot File will be saved in the directory indicated on the File Selection Menu.

Quit:

Choose this option to quit GMV.

The Display Menu

Nodes:

Nodes are points in three−dimensional space with X, Y, and Z coordinates. They may also contain various types of data such as material or velocity information.

Figure 3−1. Nodes menu

Viewing nodes, their vectors, and numbers

When the "Nodes" option is chosen from the "Display" menu, a nodes menu like Fig. 3−1 will appear. From this menu, you may choose which aspects of the node data to view. Click the "Apply" button to activate requested selections from the menu. Below the "Apply" button in the menu are three options that may be chosen individually or simultaneously. Clicking on the "Nodes" button will cause the nodes to appear as colored dots on the screen, where the colors are determined by the "Color By:" options.

Below the "Nodes" button is the "Numbers" selection box. When this option is activated, each node's respective number will be displayed next to its point on the screen.

Below the "Numbers" button is the "Color By" section. In this section you choose whether you want to color the nodes by materials, a node field, or a flag.

By default, the node data is colored according to each node's material number. However, GMV incorporates provisions to display the node data as a blue−to−red color intensity color bar according to or any user−defined field. The node can also be colored by a flag type value. To color the nodes by the current field value, click on the "Node Field:" button. To select a different field, click on the "New Field" button to pop up the Node Field Selection menu (see Fig. 3.2). To color the node by the current flag type, click on the "Flag" button. To change the current flag

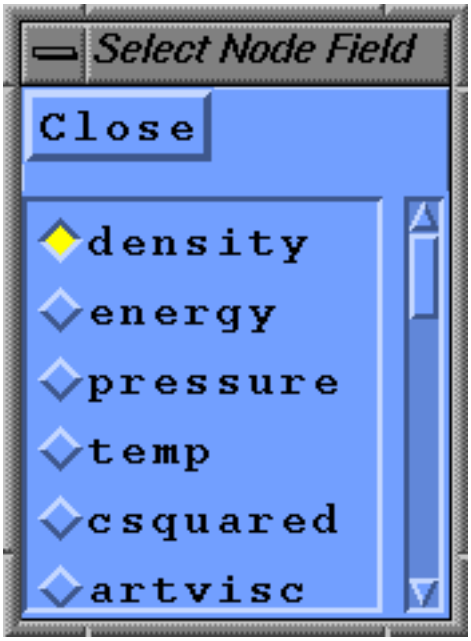


Figure 3−2. Node Field Selection Menu

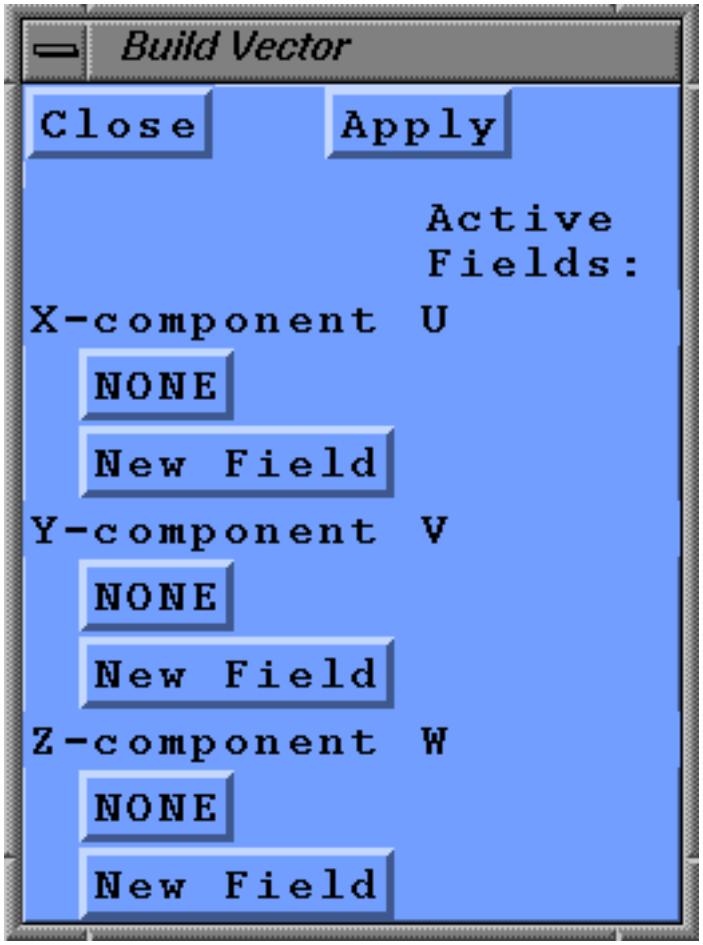


Figure 3−3. Build Vector submenu

type, click on the "New Flag" button and select a flag type from the Flag Selection menu. Nodes can also be colored by groups, if entered. If group color is selected, only those nodes that are in a group are displayed. Only one of material, node fields, flags, or groups may be selected at a time. When a node field is selected to color the nodes, a color bar scale will appear on the far left of the main viewer that gives you the range of values in relation to the color bar.

To see any nodes that have a 0 material number, or not in a group, press the "Show nodes with 0 material nos." button. Any nodes with 0 material number will have the text color as a material color.

Vectors:

Choose the "Vectors" button to build and draw vectors. To turn on the vectors, click on the button. A submenu will appear. The menu contains a toggle button to toggle the vectors on and off. When this button is selected, each node's vector data will be displayed according to the current X, Y, and Z components of the vector. Click Apply on the Nodes menu to activate vectors. The vector is colored by the "Color By" selection with a tall pyramid as the arrow head. The base of the arrow head is drawn in the text color to help determine direction.

There is also the "Build Vector" option. Click here and a submenu

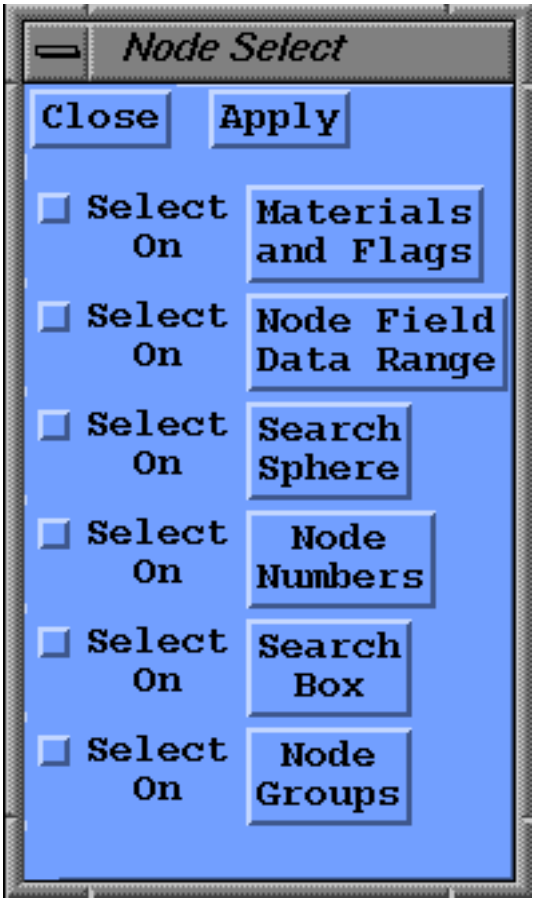


Figure 3−4. Node Select submenu

similar to Fig. 3−3 will appear. With this option, you can tailor the X, Y, and/or Z components of the vectors to any field; default is the velocity data (U, V, and W). To change a vector component, click the "New Field" button under the vector component to change. The Node Field Selection menu is then displayed (see Fig. 3−2). Choose the desired field for that vector component from the list of fields. A vector component's current field will be displayed to the right of its name under the heading "Active Fields". Choose "NONE" to zero out a vector component.

Selecting nodes to display

If you want to select certain nodes to display, click on the "Select" button in the "Nodes" menu and the "Node Select" menu is displayed (see Fig. 3−4). Use this menu to select nodes by materials and/or flags, by a data range of a node field, by a search sphere, by node numbers, a search box, or node groups. To specify a selection type, click on one of the category buttons to bring up a selection menu. After specifying the selection criteria, click on the "Select On" toggle button to use the selection type. If more than one selection type is set on, GMV will use a logical and to combine the selections. Click on the "Apply " button to process the search criteria.

Selecting nodes by Materials and Flags

To select nodes by materials and flags, click on the "Materials and

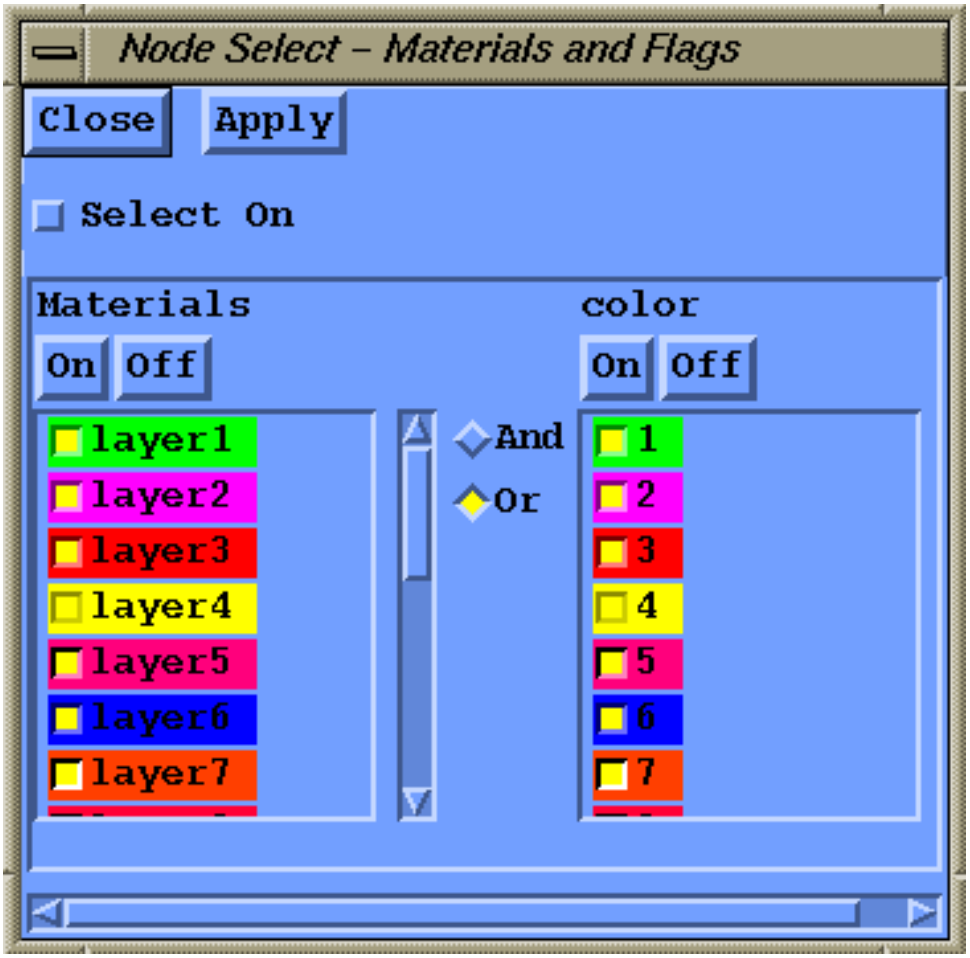


Figure 3−5. Node Materials and Flags submenu

Flags" button and the Node Materials and Flags Menu appears (see Fig. 3−5).
The available materials will be listed on the left side of the menu. Following the materials are all the flag types,if any, followed by the different flag values. Between each column of selection criteria are the two logical operators "AND" and "OR."

Highlight the desired materials, then highlight the desired flag values.
The "On" and "Off" buttons are used to select or unselect all of the materials or flags. Next, decide on the logical operator (and/or) to build a left−to−right boolean operation.

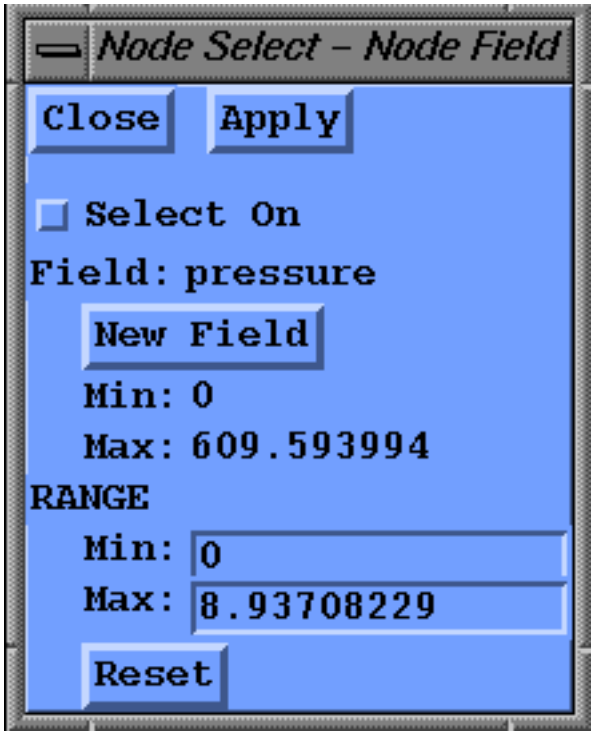


Figure 3−6. Node Field Data Range submenu

Selecting nodes by Node Field Data Range

To select nodes by a node field data range, click on the Node Field Data Range button and the "Node Field Data Range" menu appears (see Fig 3−6). Select a field to operate on by

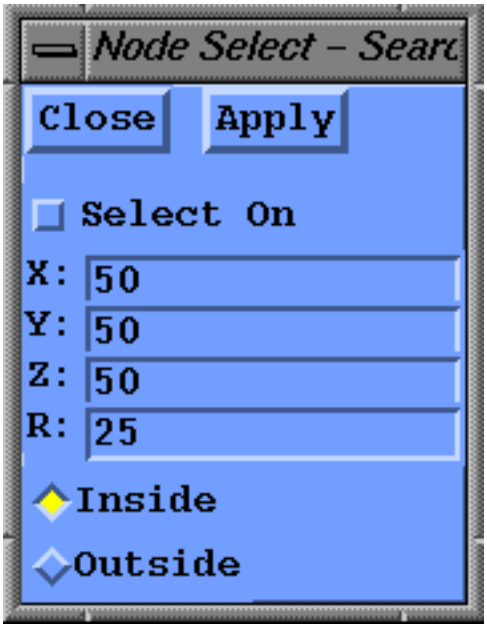


Figure 3−7. Node Search Sphere submenu

clicking the "New Field" button to pop up the Node Field Selection menu. The minimum and maximum values of the field will then be displayed. Then enter your minimum and maximum data range for node selection in the text fields. Click on the "Reset" button to reset the field minimum and maximum values in the text field.

Selecting nodes by Search Sphere

To select nodes within or outside a user defined search sphere, click on the "Search Sphere" button and the "Node Search Sphere" menu appears (see Fig. 3−7). To define the search sphere, enter the x, y, and z coordinates of the center of the sphere and the sphere's radius. Toggle the "Inside" and "Outside" buttons to select nodes in those regions.

Selecting nodes by Number(s)

To select nodes by numbers or a range of numbers, click on the "Node Numbers" button and the "Node Numbers" menu pops up (see Fig 3−8). The menu contains 50 lines where individual node numbers, a range of node numbers, or a range of node numbers with a stride can be entered (one entry per line). A colon (:) is used as the delimiter when defining a range of numbers or a range of numbers with a stride. The format used to define a range of node numbers is first:last, e.g. 1:10. The format used to define a range

Figure 3−8. Node Number submenu

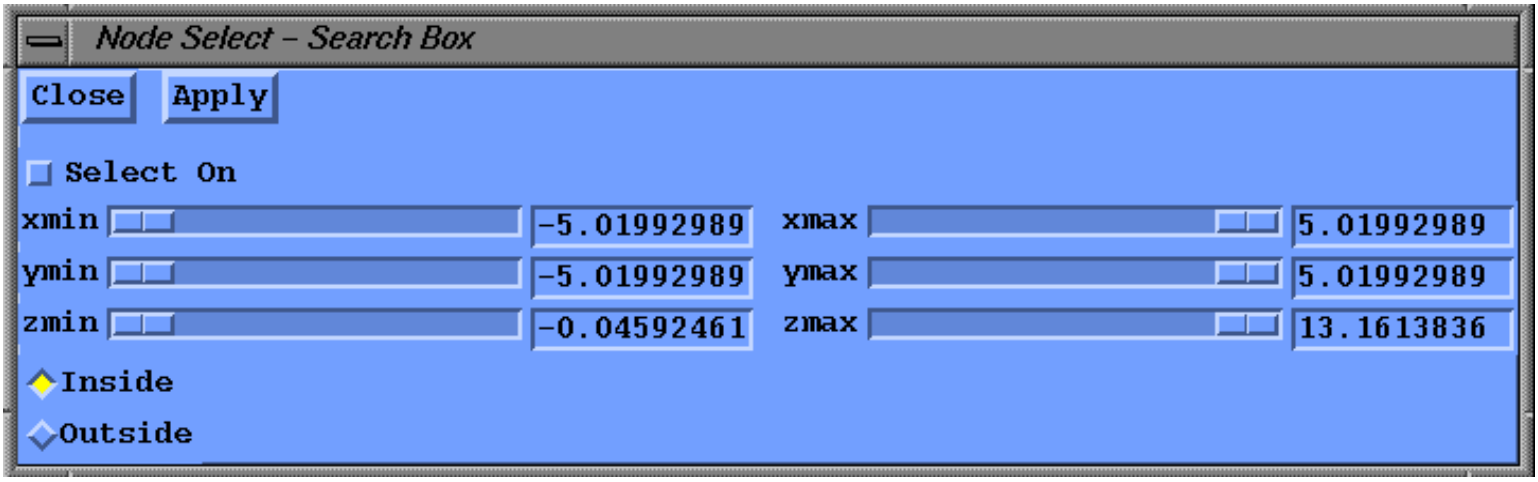


Figure 3−9. Node Search Box submenu

of node numbers with a stride is first:last:stride, e.g. 20:100:10. All node numbers must be greater than 0, and any number greater than the maximum node number will be reset to the maximum node number. If a line contains an invalid character, an error message will appear in the Node Select menu indicating the line number with the error.

Selecting nodes by Search Box

To select nodes within or outside a user defined search box, click on the "Search Box" button and the "Node Search Box" (Fig. 3−9)menu appears . To define the search box,either enter the xmin, ymin, zmin and xmax, ymax zmax which define the coordinates of two points at opposite corners of the box, or move the sliders to interactively define the box. When the sliders are moved, a green box appears that defines the search box. Toggle the "Inside" and "Outside" buttons to select nodes in those regions.

Selecting nodes by Groups

To select nodes defined in a group, click on the "Node Group" button and the "Node Select Node Group" menu appears . Use the left mouse button to select groups. Click on the left mouse button to select a single group, hold the left mouse button down and drag the mouse to select a block of groups. The shift key can also be used to select a block of groups. The Ctrl key and the left mouse button can be used in combination to toggle a group on/off. Toggle the "Inside" and "Outside" buttons to select nodes in those groups.

Activating node selection

Be sure to click the "Select On" toggle button in the Node Select menu or on the specific selection menu. Then click on the "Apply" button to start the selection process.

Cells:

Cells are three−dimensional fixed shapes in space, such as cubes, pyramids, or prisms. Cells are defined by the nodes at their vertices in the input file.

Viewing cell faces, edges, and numbers

Choosing the "Cells" option under the "Display" menu brings up the "Cells" menu, (Fig. 3−10). Here you may choose which aspects of the cells you wish to view. The "Faces" option displays the face of the cells as an interpolated blended color polygon where the colors are based on the selected cell or node field. If the cell face is colored by a node field, the "Contours" option under "Faces" displays contour lines on the faces. The contour lines are drawn at the intervals shown on the cells Color Bar. The "Shaded" option under "Faces" turns on the lighting model and shades the faces. The "Refine" option, available only when the cell faces are colored by a node field, adds an interpolated point at the face center to provide a smoother color change across the face. When the "Test Normals" option is selected, cells with inward pointing normals will be drawn in black (outward face normals using the righthand rule is the standard). The "Edges" option displays the edges of the cells as a colored wireframe image. The "Median Mesh Edges" option generates and displays median mesh edges. If both "Faces" and "Edges" are selected, the edges are colored grey. If both "Edges" and Median Mesh Edges" are selected, cell edges are drawn in black. If "Faces", "Edges", and Median Mesh Edges" are selected, the median mesh is drawn in white. The "Cell Numbers" option draws the cell number at the center of the cell in the edge color for the cell while the "Node Numbers" option draws the cell's node numbers in the text color and the "Face Numbers" option draws the cell's face

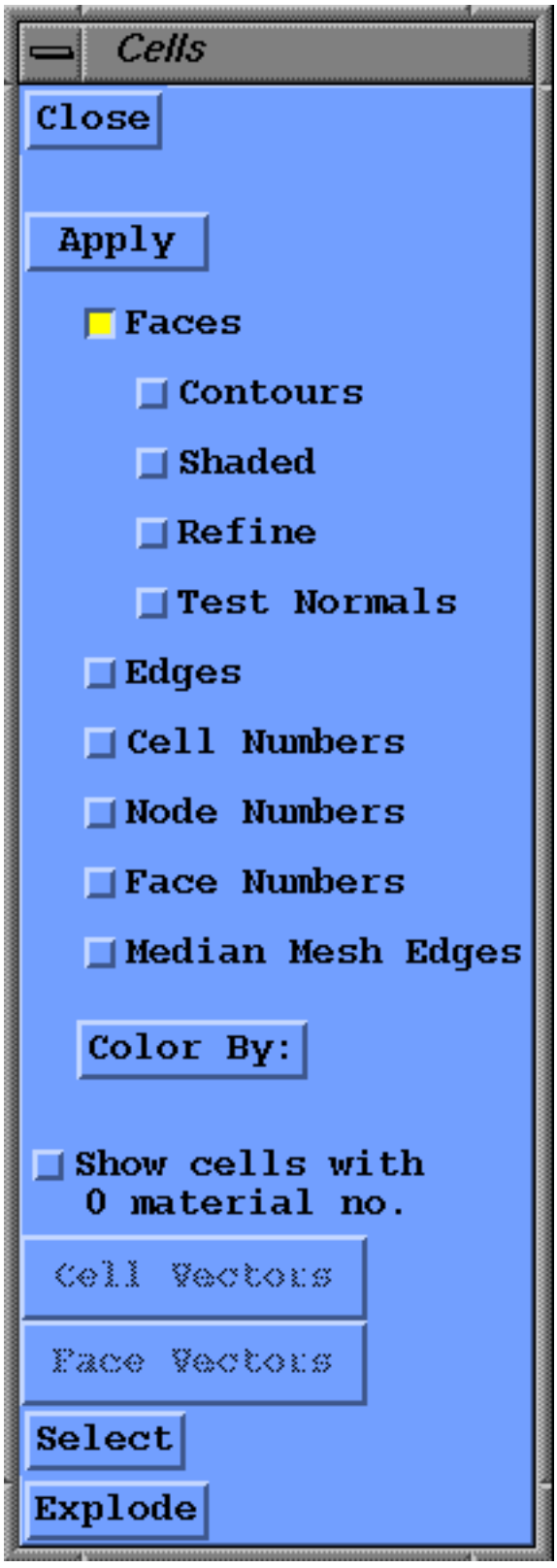


Figure 3−10. Cells menu

numbers in the text color. Cell faces, edges, and numbers may be displayed simultaneously. Simply click on the box next to the desired option just as is done in the "Nodes" menu. Click on the "Apply" button to activate the selections.

Coloring cells by materials, fields, and flags

Click on the "Color By:) button to display the "Cell Color By" submenu (see Fig. 3−12). Cell edges are colored by material color, flag values, or a blue−to−red intensity color for cell− centered, node−centered or face−centered field values. If node field values are selected the edges are smoothly colored. The cell faces can be colored by material or flag values as well as a blue−to−red intensity color of cell−centered or node−centered field values. Cell faces can also be colored by cell groups or face groups, only cells or faces that are in a group are displayed.

To see cells that have a 0 material number, or not in a group, press the "Show cells with 0 material no." button. Any cells with 0 material number will have the text color as a material color.

Cell vectors

If there is cell−centered data, cell vectors can be built. Click on the "Vectors" button to bring up the appropriate submenu. The procedure to display and build vectors is the same as node vectors.

Face vectors

If there is face−centered data, cell vectors can be built. Click on the "Vectors" button to bring up the appropriate submenu. The procedure to display and build vectors is the same as node vectors. Face vectors are drawn in a grey color.

Selecting cells to display

This is done in the same manner as described in the previous section about nodes and their materials and flags, cell field range, search sphere, cell number(s), search box, and cell groups. Additionally, materials can

Figure 3−11. Cell Color By submenu

be selected or removed interactively from the screen. The "Materials and Flags" selection menu includes the "Select Material From Screen" and "Remove Material From Screen" toggle buttons (see Fig. 3−12). Clicking on the "Select Material From Screen" button will turn off all the material buttons and a crosshair cursor will appear on the display. Click the left mouse button on a cell in the display and the material button for which the cell is part of will be turned on. Clicking on the "Remove Material From Screen" button will display a crosshair cursor. Click the left mouse button on a cell to turn off its material button. As before press the "Apply" button to display the selected cells.

Cells can also be selected by a node field range for the nodes that define the cells. The "Cell Node Field Range" menu contains two additional buttons, the "Any" and the "All" toggle buttons. The "Any" button will select a cell if any of its nodes fall between the node range valued. If you want to select cells with all nodes falling between the node field range, click on the "All" button. If face groups are selected, only selected faces are drawn.

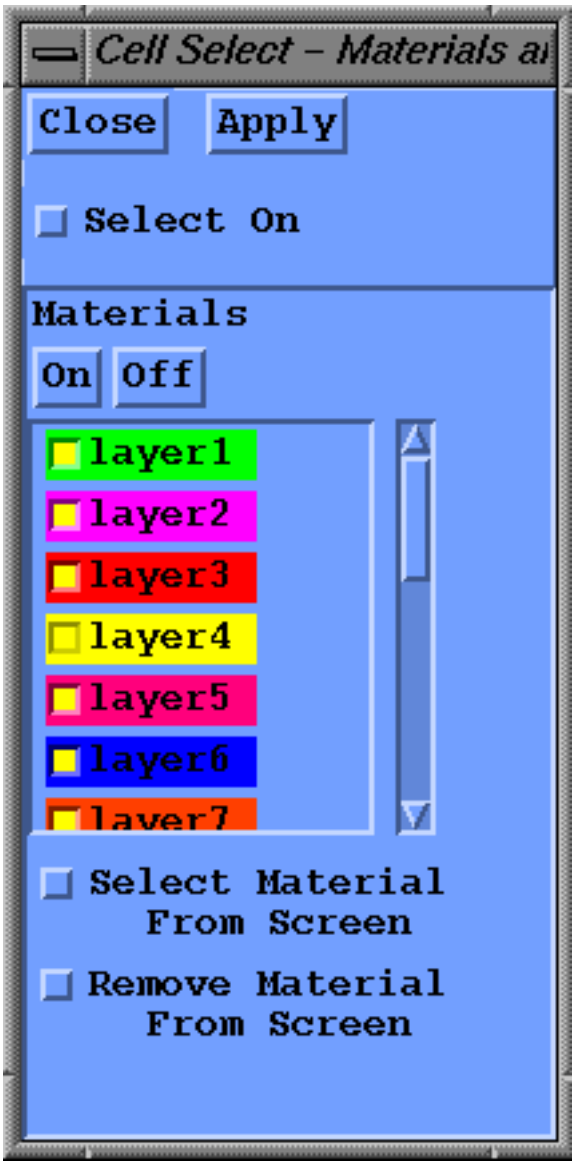


Figure 3−12. Cell Materials and Flags submenu

Explode

Explode allows you to separate groups of cells based on material or flag data. Clicking on Explode pops up the Cell Explode submenu, (see Fig. 3−13). Use the "Explode %" slider to adjust the amount of separation of cell groups as a percentage of the distance. Use the "Explode On" radio buttons to select the material number ("Mat. No") or flag type to group the cells. Click on "Apply" to display the separated groups of cells. To return to the initial setup, return the slider bar back to zero and press apply.

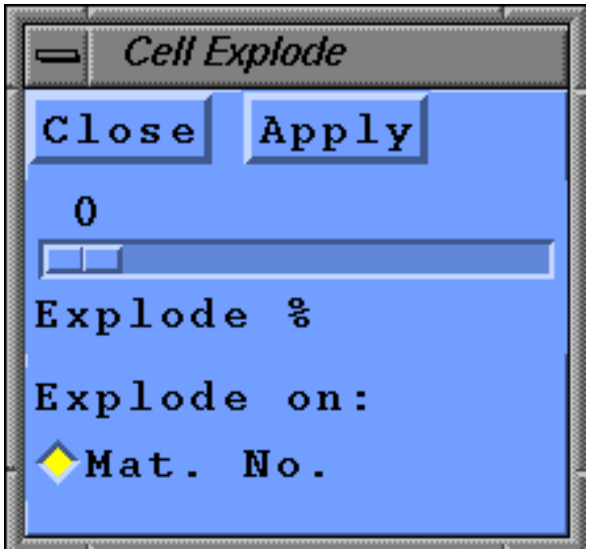


Figure 3−13. Cell Explode submenu

Polygons:

The "Polygons" menu is used to view the surfaces made by the input polygons, (see Fig. 3−14). Polygons are surface facets that are shaded according to the location of the light source. The polygons are colored by material color as specified in the input file.

Shading and outlining polygons

To change the way GMV displays polygon data, choose the "Polygons" option from the display menu. The "Apply" button is used to activate
the desired selection from this menu. The polygons can be outlined, shaded, or both, depending on your preference. The way the polygons are "Shaded" depends on the location of the light source. This location of the light source can be changed by using the light source box in the upper right corner of the main GMV window. See the information on page 1−7.
When the "Lines" option is selected, GMV draws lines between the vertices of the polygon, to create a wireframe image. If both "Shaded" and "Lines" options are selected, the polygon lines are colored white.

Selecting materials to display

The surface materials may be turned on or off. The materials are listed at the bottom of the "Polygons" menu. For convenience, there is an on and off button to turn all the materials on or off at once. Use the material toggle buttons to select individual material surfaces for display. The "Select Material From Screen" and "Remove Material From Screen" toggle buttons allow interactive selection as in the Cell Material selection Menu. Press the "Apply" button to activate the selection.

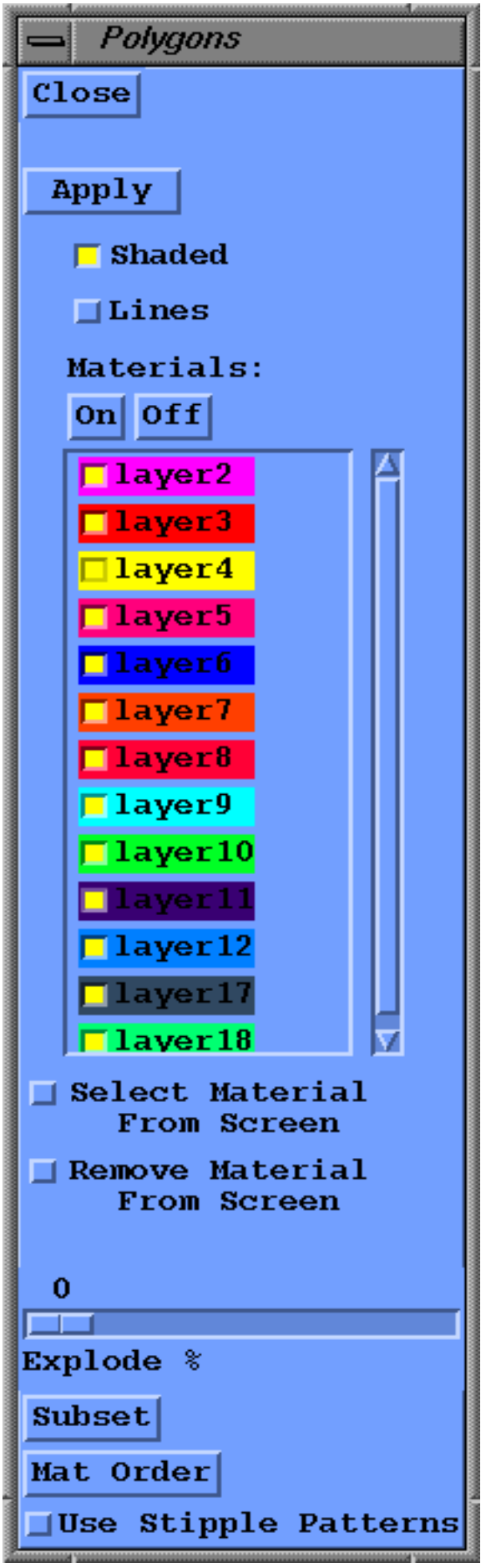


Figure 3−14 Polygons menu

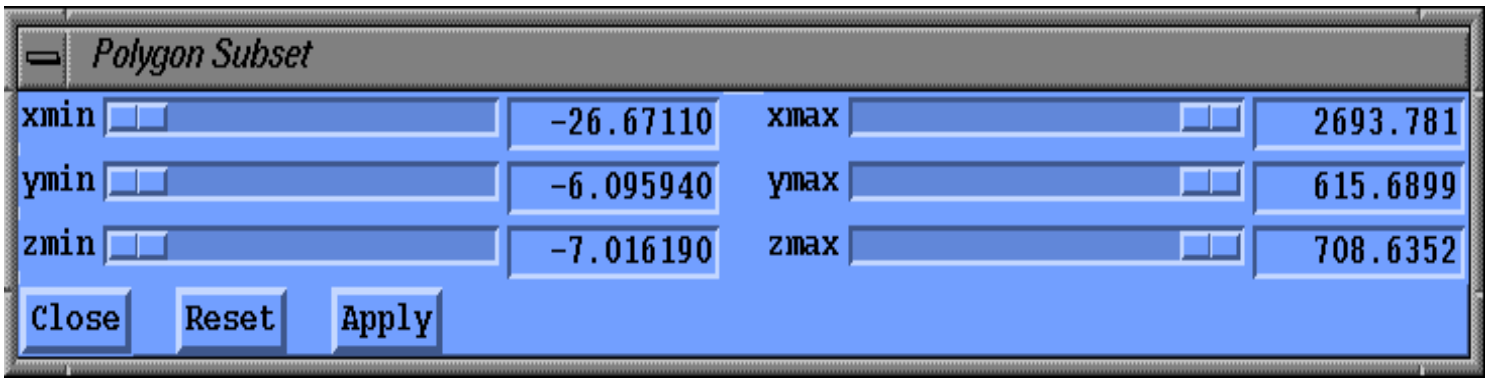


Figure 3−15. Polygon subset submenu

Changing explode percentage

Use the "Explode%" slider to separate surfaces by material, giving an exploded view of the surfaces.

Selecting a polygon subset

When you click on the button labeled "Subset" in the "Polygons" menu, a submenu labeled "Polygon Subset" will appear, (see Fig. 3−15). There are six sliders and six text boxes in the window. These are used to define a polygon subset region. The minimum and maximum values listed in the six text boxes define a cube in space. Any polygon that does not lie completely within this cube will no longer be shown in the main viewer. Use the sliders to modify the polygon subset cube. A purple bounding box is displayed as the sliders are moved to help visually set the polygon subset region. Alternatively, a value can be entered in the text portion of the menu. Press the "Apply" button to initiate the subset. Any polygon not falling within the new polygon subset region will be erased. Clicking on the close button will close the subset window, leaving any changes in effect. Clicking on the Reset button will reset the polygon subset box to the problem size and all polygons will then be drawn.

Changing material order

The material order function comes into play when two or more polygons occupy the same region of space. Because only one of the stacked polygons may be shown at a time, you must decide the order of precedence for the materials so that GMV will know which polygon to draw. Material order is also important when more than one material is transparent. Transparent materials must be in a back−to−front order in order to be drawn correctly. By default, material number 1 is first, followed by material number 2 and so on.

To change the current material order, click on the button labeled "Mat Order" in the "Polygons" menu. A submenu similar to Fig. 3−16 will appear. On the far right you will see the current materials listed in order. To change the order, click on each material button from the left−hand column in the desired order. The

Figure 3−16. Material Order submenu

materials will be listed in the middle column in the proposed order. Each material may be chosen only once. If you make a mistake in the material ordering process, click on "Undo." This will clear the order listing of materials and allow you to start over. When the materials are in the desired order, click on "Apply" and the changes will take effect. The new order number will appear on the right−hand column. Click on "Close" when finished.

Tracers:

Tracers are points in space used for monitoring data in locations where nodes do not exist. Tracers are defined by their X, Y, and Z coordinates and by the data they are assigned by the input file. Tracers can be assigned multiple fields of data, such as in Fig. 3−17 where it reads pressure and temperature data from the input file.

Methods of displaying tracers

Tracers can be represented in four different ways by GMV, either as regular points, big points, numbers, or values. When the "Tracers" menu is brought

up from the "Display" menu bar, you will notice four selection boxes under the heading "Draw as." When the "None" option is chosen, no tracer data will be shown. The "Regular Points" option displays tracer data as colored points whose size is determined by the "Point Size" option in "Controls 2". The "Big Points" option displays tracer data as large, bright icosahedra centered on the tracer location. When "Numbers" is selected, the sequential number of each tracer is displayed. Do not get the tracer numbers confused with the node or cell numbers. When all three are displayed simultaneously, it is hard to tell them apart. Finally, there is the "Values" option. This one tells GMV to display the value of each tracer in the correct location on the screen. Depending on which field is currently selected, the colors for tracer display are a blue−to−red intensity color depicting the values of the field selected for display. Click on the "Apply" button to activate your selections.

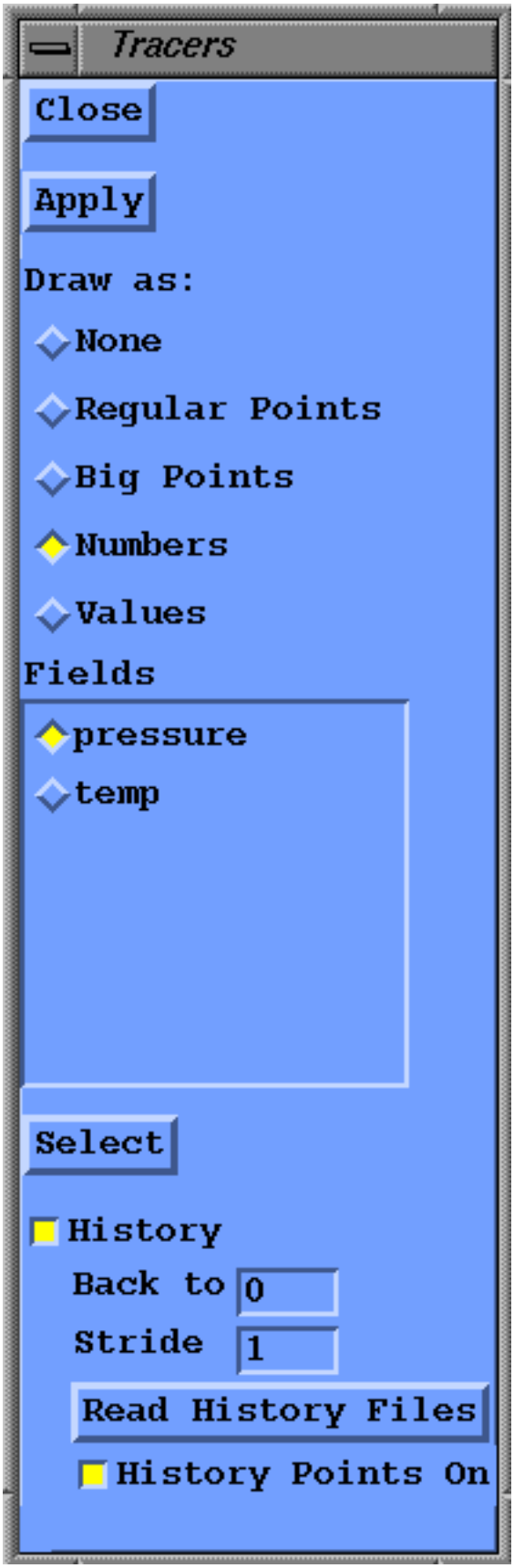


Figure 3−17. Tracers menu

Selecting data field for tracer to represent

A main color bar pops up inside the GMV main viewer when tracers are displayed. The tracer always takes on the color of its value in the currently selected field. The available fields are listed under the heading "Fields" at the bottom of the tracer window. To select a field, click on the radio button next to its name. The colors of the tracers in the main viewer will change. If the input file read in by GMV contained no tracer data, then there will be no fields to choose from in the "Tracers" menu. Selecting the "Close" button will close the menu, leaving any changes you have made intact.

Selecting tracers to display

Tracers can be selected for display by number(s). Click the "Select" button to display the "Tracer Select Menu" (see Fig. 3−18). As in selecting nodes or cells by number, there are 50 lines where a tracer number, a range of tracer

numbers, or a range of tracer numbers with a stride can be entered. Use a colon (:) as the delimiter for a range of numbers and for a range of numbers and a stride. For example to select tracers between 1 and 10, enter 1:10, to select every tenth tracer between 20 and 100, enter 20:100:10.

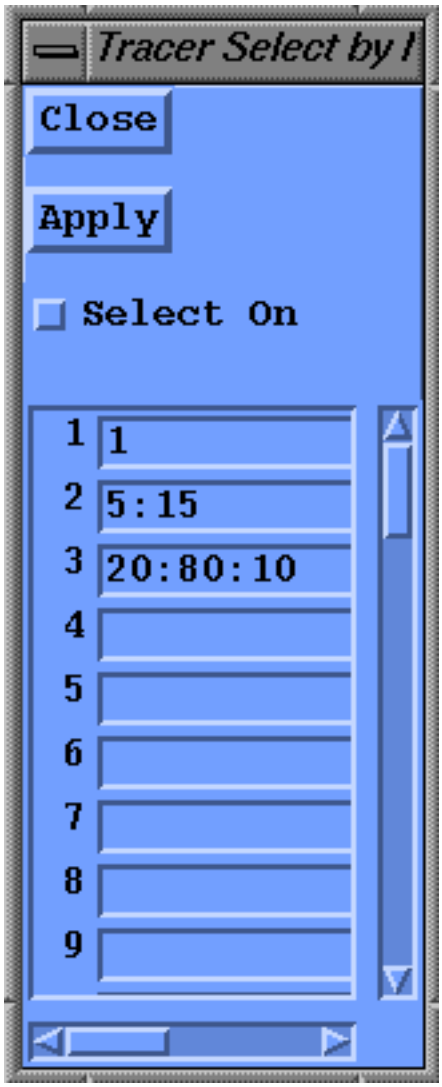


Figure 3−18. Tracer Select submenu

Display tracer history

Tracer histories are displayed as a set of colored line segments connecting with the big points, numbers or values display of the current tracers. The line segments connecting the history locations are smoothly colored according to the value of the selected field for up to 250 history points per tracer. Click the "History" button to turn on the tracer history display option and click the "Apply" button to initiate the option. The "Back to" and "Stride" input areas are used to control the last file to read and a stride (skip value) between files. Click on the "Read History Files" to read trace histories. Click on the "History Points On" to toggle drawing of intermediate history points.

In order to read trace histories, all of the GMV input files must have the same file name prefix and must have a 3 or 4 digit number as the file name suffix. GMV will only read history files that exist in the directory of the current GMV file and will continue to read the files until there are no more files to be read or until 250 files have been read. Once a set of history files are read, GMV will not read any more trace histories until another GMV input file is read using the Read GMV option of the File menu.

Surfaces:

Surfaces are sets of facets defined by mesh nodes at their vertices in the input file. Surface facets can have their own material numbers, flags, velocities and field data.

Viewing surface faces, edges, and numbers

Choosing the "Surfaces" option under the "Display" menu brings up the "Surfaces" menu, (Fig. 3−19) Here you may choose which aspects of the surfaces you wish to view. The "Faces" option displays the surface facet as an interpolated blended color polygon where the colors are based on the selected surface or node field. If the surface face is colored by a node field, the "Contours" option under "Faces" displays contour lines on the faces. The contour lines are drawn at the intervals shown on the surfaces Color Bar. The "Shaded" option under "Faces" turns on the lighting model and shades the facets. The "Refine" option, available only when the surface facets are colored by a node field, adds interpolated points at the facet center and edge centers to provide a smoother color change across the facet. The "Edges" option displays the edges of the surfaces as a colored wireframe image. If both "Faces" and "Edges" are selected, the edges are colored grey.. The "Surface Numbers" option draws the surface number at the center of the facet in the edge color for the surface while the "Node Numbers" option draws the surface's node numbers in the text color. Surface faces, edges, and numbers may be displayed simultaneously. Simply click on the box next to the desired option just as is done in the "Nodes" menu. Click on the "Apply" button to activate the selections.

Coloring surfaces by materials, fields, and flags

Surface edges are colored by material color, flag values, or a blue−to−red intensity color for surface− centered or node−centered field values. If node field

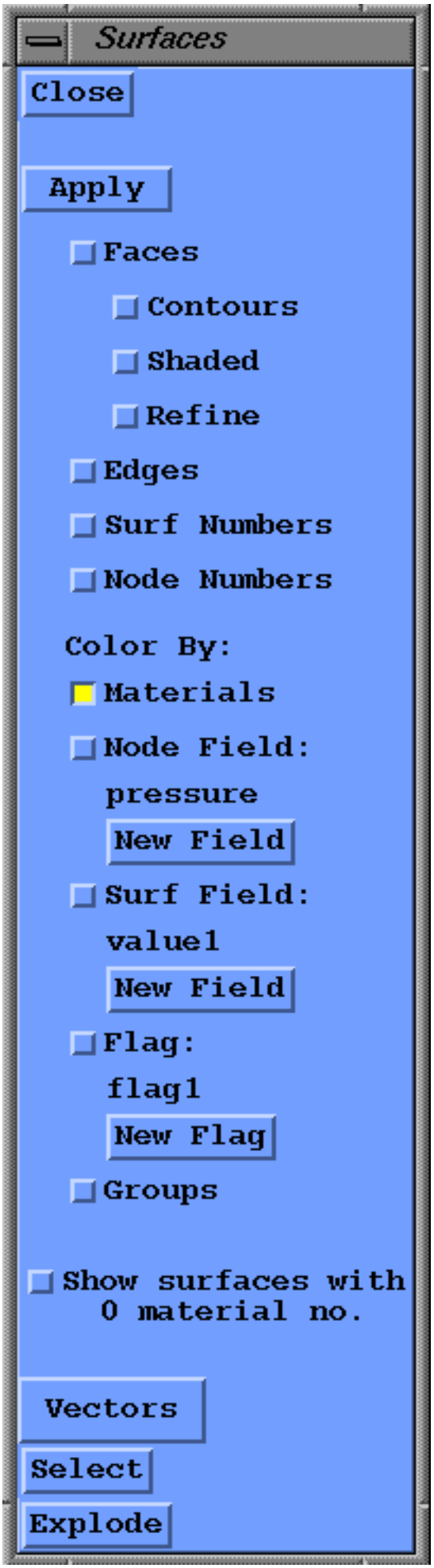


Figure 3−19. Surfaces menu

values are selected the edges are smoothly colored. The surface faces can be colored by material or flag values as well as a blue−to−red intensity color of surface−centered or node−centered field values. Surfaces can also be colored by groups, only surface elements that are in a group are displayed.

To see surfaces that have a 0 material number, or node group, press the "Show Surfaces with 0 material no." button. Any surfaces with 0 material number will have the text color as a material color.

Surface vectors

If there is surface−centered data, surface vectors can be built. Click on the "Vectors" button to bring up the appropriate submenu. The procedure to display and build vectors is the same as cell vectors.

Selecting surfaces to display

This is done in the same manner as described in the previous section about cells and their materials and flags, surface field range, node field range, search sphere, surface number(s), search box, and surface groups.

Explode

An exploded view of surfaces is accomplished just as in cell explode.

The Calculate Menu

Average:

The "Average" function in the "Calculate" menu calculates a weighted average, by material, of two node fields or cell fields. Chose the "Average" option from the "Calculate" menu and select either "Node Fields" or "Cell Fields" (if available) and a menu similar to Fig. 4−1 appears.

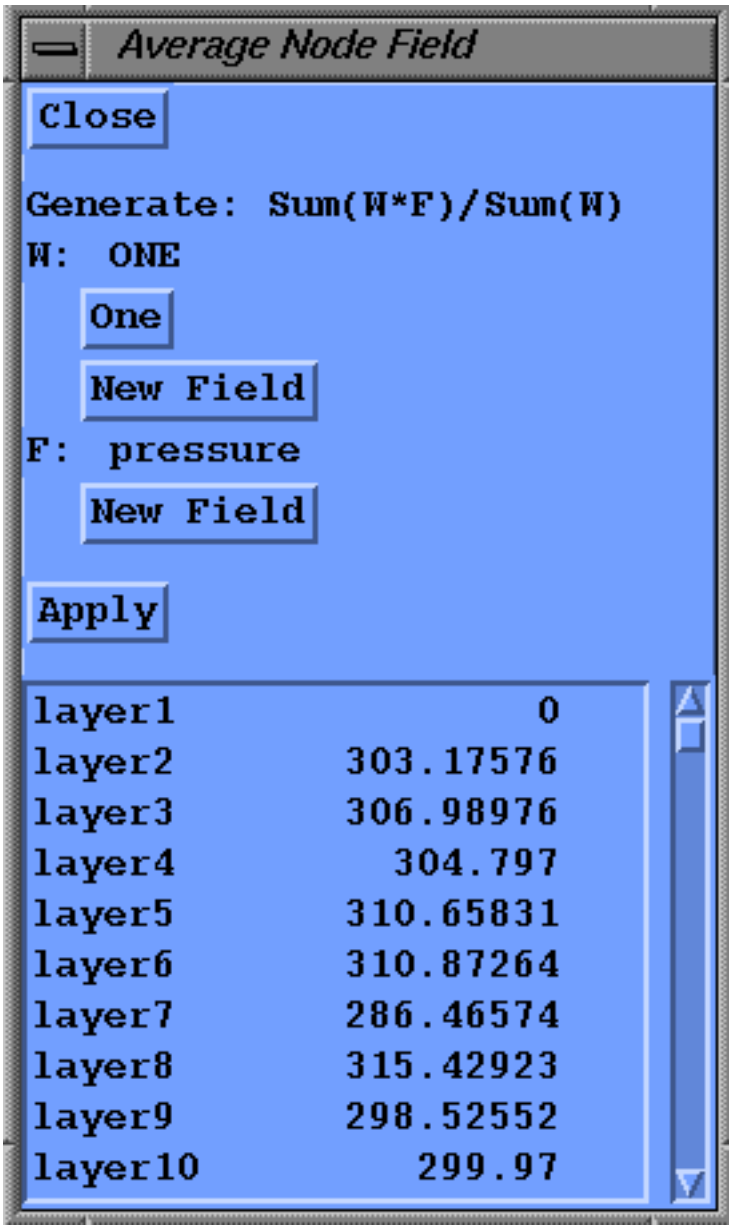


Figure 4−1. Average menu

Selecting a weighting field

To select the field to be used as the weighting function, click on the "New Field" button under the W: designation. For an unweighted average, click on the "One" button.

Selecting a field to Average

To select the field to average, click on the "New Field" button under the F: designation. Then click on the "Apply" button to generate the averages. The averages will be displayed in the scrolling list under the "Apply" button.

Cutlines:

The "Cutlines" function in the "Calculate" menu allows up to twenty cutlines to be generated. A cutline is the centerline of a cylinder with a user defined radius. The field value of any node that lies within the cylinder is projected onto the cutline and displayed as a blue−to−red color−coded line.

Selecting a cutline

Choose the "Cutlines" option from the "Calculate" main menu bar and a menu similar to Fig. 4−2 will appear. The numbers are the cutline number and "NONE" indicates that the cutline has not been generated. A field name indicates that a cutline has been generated for that field. Select one of the cutlines and a menu similar to Fig. 4−3 will appear.

Creating a cutline

A cutline can be created in one of five methods. The line can be defined by entering the x, y, and z values of the two endpoints of the line. These can be entered on the two rows of boxes identified as "P1" and "P2". A second method is to use the "1 Point" option. Click on the "1 Point" box then move the cursor to the main viewer. A crosshair cursor will appear; place the cursor on a point on the screen, and

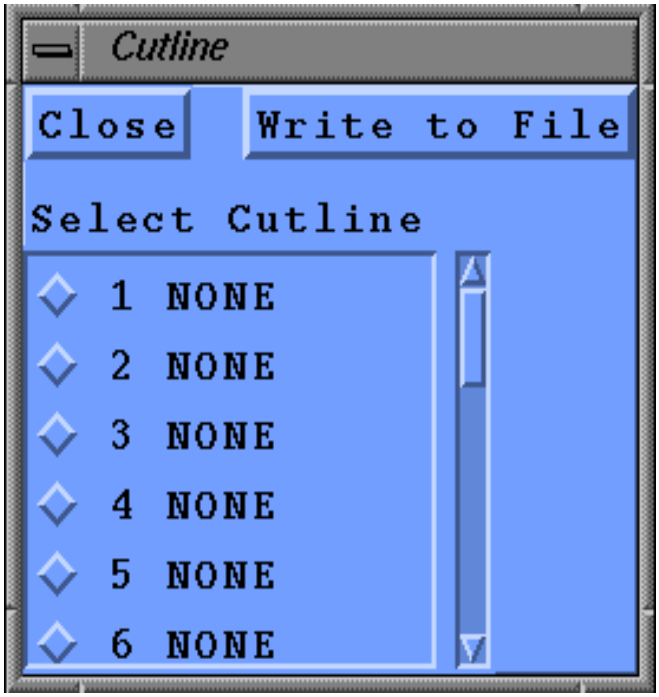


Figure 4−2. Cutline Selection menu

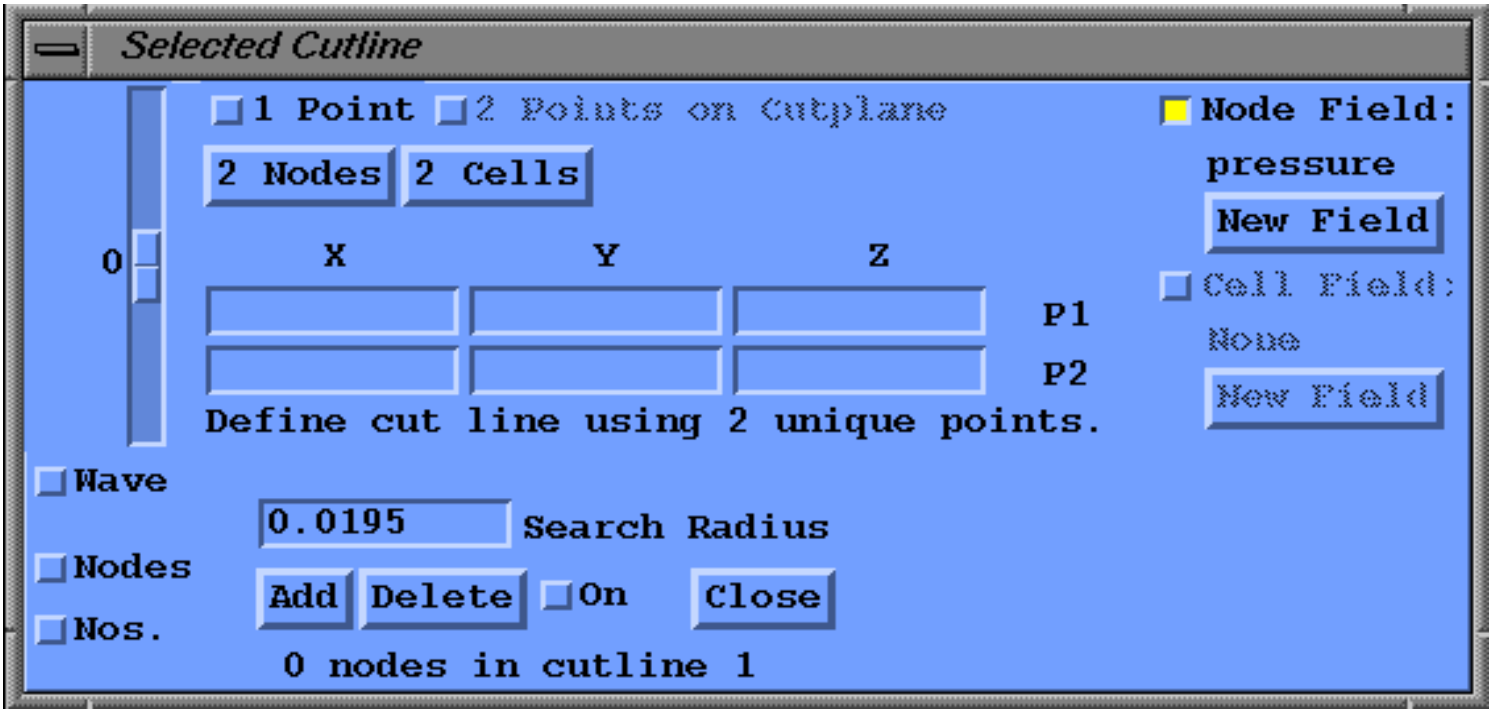


Figure 4−3. Create cutline menu

click the left mouse button. A line will be generated normal to the screen with endpoints at the plot box intersections with the line. The third method of defining the line is with the "2 points on Cutplane" option. If a current cutplane exists, select the "2 points on a Cutplane" option and move the cursor to the main viewer. Move the crosshair to a point on the cutplane and click the left mouse button; then move to the second point on the cutplane and click the left mouse button. A line will be defined where lines normal to the screen from the selected points intersect the cutline. It is possible to select the "2 points on a Cutplane" option when multiple cutplanes are defined. To avoid ambiguity, all cutplanes other than the desired one should be disabled prior to creating the cutline with this option.

Press the "2 Nodes" or "2 Cells" buttons to display a menu where tow node or cell numbers can be entered. Click the "Apply" button and the coordinates of the nodes or cell centers will fill in the two point values.

When the cutline has been defined, enter the cylinder radius in the "Search Radius" box if the default radius is not appropriate. Hint: Use the "Distance" option of the "Calculate" menu to determine an appropriate distance. Next, select a field whose node or cell data will be color coded along the line. Finally, click on the "Add" button to create the cutline; a message will then be displayed along the bottom of the menu showing the number of nodes selected for this cutline. The corresponding button in the Cutplane Selection will display the field value selected.

Cutline display options

The "On" button toggles the cutline display for the selected cutline on or off. The "Nodes" button, when on, displays the nodes selected for the cutline. The "Nos." button displays the node numbers for the nodes selected for the cutline. The "Wave" toggle button with its slider is used to display the function wave of the data along the cutline. Use the wave slider to increase/decrease the amplitude of the wave. The "Delete" option deletes the current cutline, deleting a cutline causes the corresponding button in the Cutline Select menu to display "NONE".

Cutline 2D plot

When cutlines are generated, a 2D plot of the cutlines appears. The plot automatically scales all cutlines and stays visible until all cutlines are deleted, or the main cutline menu is closed.

Cutplanes:

A cutplane is a plane through the simulation onto which data is interpolated. GMV will display only the parts of the cutplane that intersect with the mesh data in the main viewer. Cutplanes are useful for generating color

contour plots of data for detailed analysis.

Main Cutplanes Menu

The Main Cutplanes Menu is accessible by choosing the "Cutplane" option from the "Calculate" main menu bar. This menu, shown in Fig. 4−4, contains several buttons that allow the specification of how a field is interpolated by a cutplane. Descending the menu, the first button allows the retrieval of a field value from an arbitrary location on a cutplane. The next set of buttons allow the selection of creating the cutplane based on node or cell data, and the particular field dataset to be used in each case. In addition to the ability to select nodes or cells for the desired field data type to display, this menu allows the modification and update of field variables displayed on existing cutplane(s) (in this case, changes affect all active cutplanes simultaneously). This menu allows the selection of particular cutplanes with the use of the toggle buttons in the lower portion of the menu.

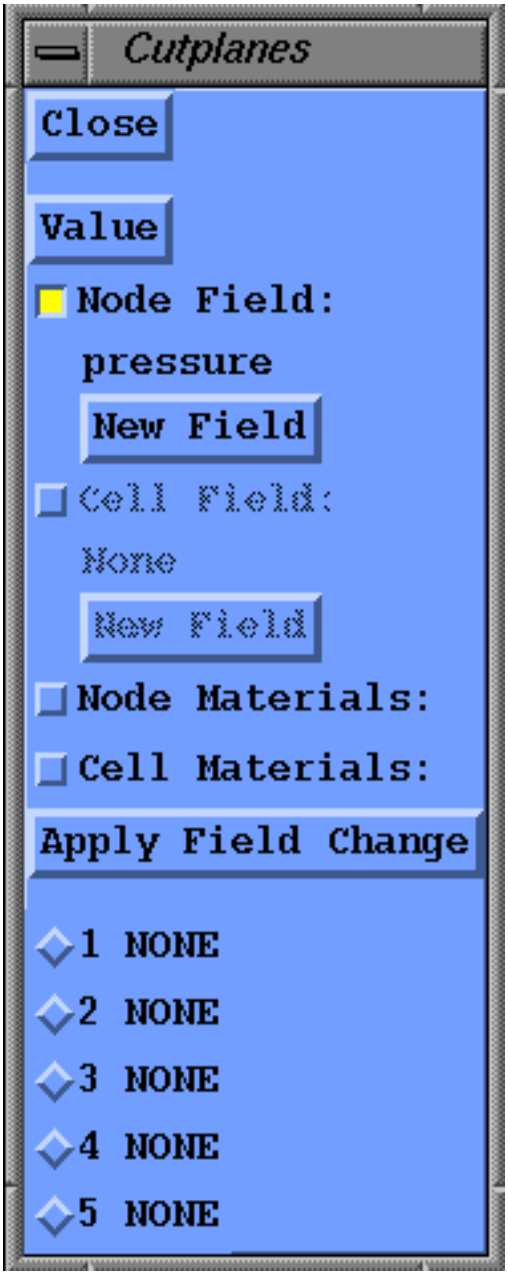


Figure 4−4. Main Cutplanes menu

Value

The "Value" button is located in the upper left corner of the "Cutplane" menu. The "Value" button can be used to determine the contour value at any point along the cutplane. To determine a contour value, first click on the "Value" button. Move the mouse pointer to the main viewer, at which point it will change into crosshairs. Click the crosshairs anywhere on the cutplane. The value at that point will then be displayed along with the current field name to the right of the "Value" button.

Node Field, Cell Field or Material

The "Node Field", "Cell Field" "Node Material" or "Cell Material" toggle buttons select which type of data to be used to color the new cutplane. In the

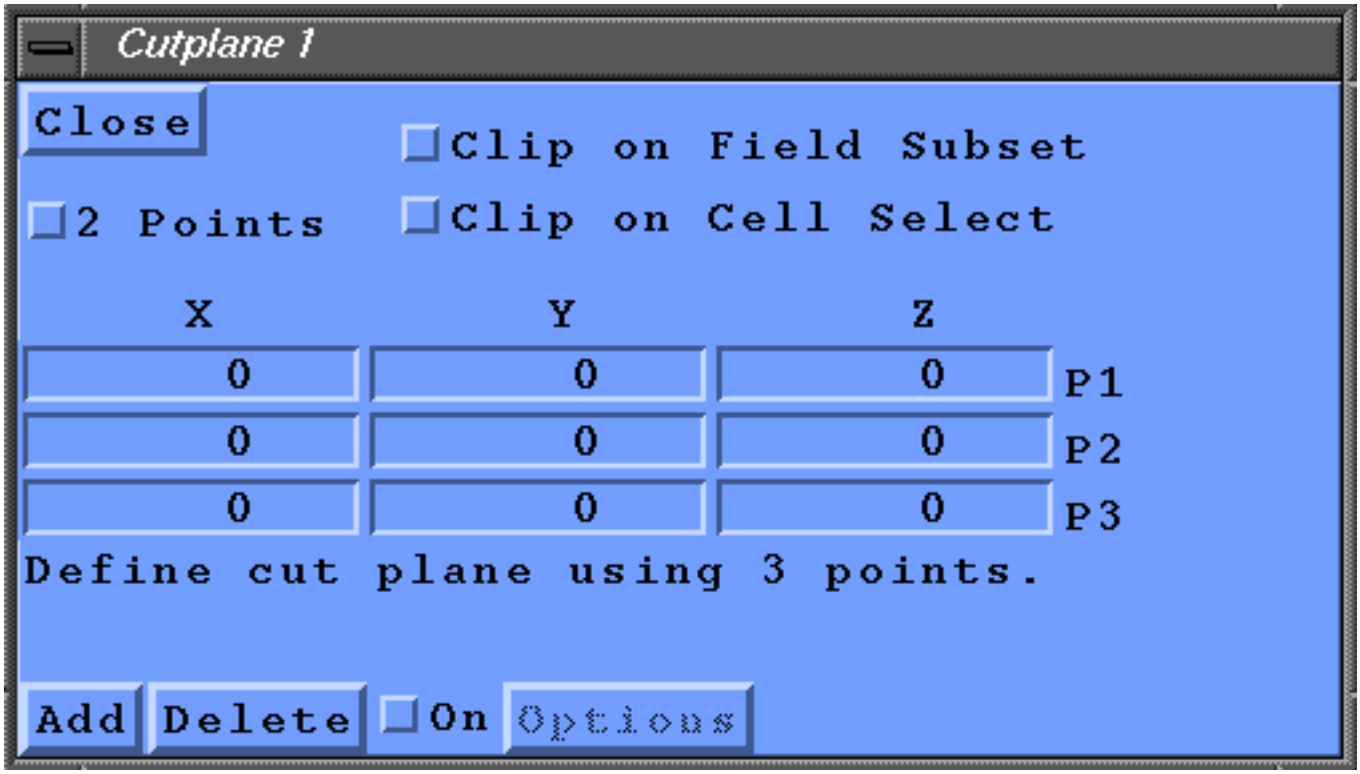


Figure 4−5. Cutplane menu

case of the modification of existing cutplanes, these toggles allow changing the current data used to color the cutplanes.

New Field

The "New Field" button under both of the "Node Field" and "Cell Field" selectors allows the specification of the particular data field desired when node or cell data is selected. This feature may also be used to modify the field used on an existing cutplane.

Apply Field Change

The "Apply Field Change" button updates the defined cutplane(s) with any changes made to the field or material configuration. If multiple cutplanes are defined, field or material changes will be applied simultaneously to all cutplanes.

Cutplane Selection Buttons

The last five buttons in the "Main Cutplanes Menu" allow the selection of particular cutplanes. When a new cutplane is added, the uppermost unused toggle is selected to define the cutplane and specify its location. The selection of an unused toggle generates a menu similar to Fig. 4−5 for cutplane creation. When a cutplane has been specified and created, the "NONE" tag in the main menu next to the selection button changes to "ON." If the cutplane is subsequently toggled off, the "ON" tag changes to "OFF." If the cutplane is deleted, the tag changes back to

"NONE." Thus, it is evident which cutplanes are active and how many have been created, in addition to providing the capability to specify a particular cutplane.

Cutplane Description Menu

When a particular cutplane from the lower portion of the main menu is selected, the description menu (Fig. 4−5) is created. This menu incorporates the "2 Points" selection button and the data fields where cutplane coordinates may be entered as described above.

Clip on Field Subset and Cell Selection

The two "Clip" toggle buttons, found at the top of the "Cutplane Options" submenu, allow you to clip the cutplane according to the subset defined in the subset tool if you choose "Clip on Field Subset", or according to the subset defined by the Boolean expression created using the "Select" option in the "Cells" menu if you choose the "Clip on Cell Select." When you select one or more of these buttons, the cutplane will only be drawn from data within the limits of the subset, leaving out everything outside the subset. These buttons must be selected before adding a cutplane in order for them to take effect.

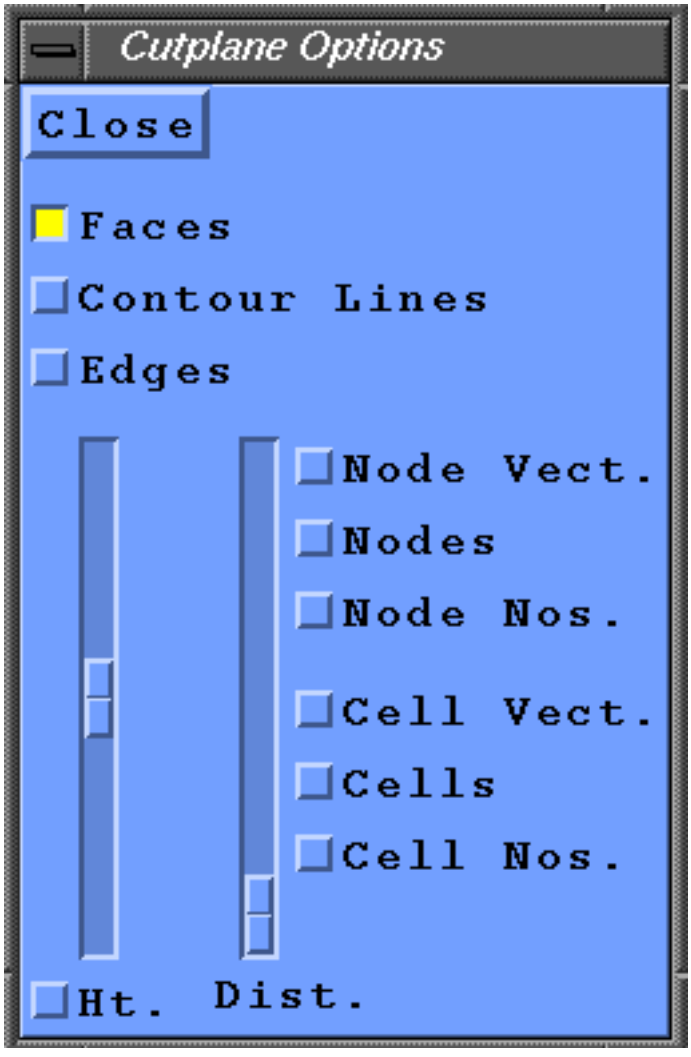


Figure 4−6. Cutplane Options menu

Cutplane Options

The button labeled "Options" in the cutplane menu pops up the submenu called "Cutplane Options." In the cutplane options submenu (Fig. 4−6) there are two slider bars and several toggle buttons. Cutplane option changes apply only to the currently selected cutplane.

Faces

One of the toggle switches in the "Cutplane

Options" submenu is an option labeled "Faces." This toggle controls the visibility of the colored polygons that compose the cutplane. Turning "Faces" off allows better visibility of contour lines and edges if these options are selected.

Contour Lines

Below the "Faces" toggle button is a button labeled "Contour Lines." When selected, this button will draw contour lines in the current text color (either black or white depending on the background color) on the cutplane corresponding to the same divisions found in the color bar. For example, if the color has divisions at 5, 10, and 15, then GMV will draw contour lines at these intervals.

Edges

If you want GMV to draw the edges of any cells the cutplane intersects, click the mouse on the "Edges" button. There is no need to make sure the option is checked before adding the cutplane. The edges of the intersected cells will instantly appear.

Height

The "Height" slider bar allows you to make your cutplane appear three dimensional for greater clarity. The slider bar moves each point on the cutplane away from the plane according to its value as indicated by its color. For example, the red areas, which are the most intense, move away from the plane the greatest distance while the blue areas move the least. To activate the "Height" feature, click on the toggle button on the lower left corner of the "Cutplane" menu. Now drag the slider up and down to adjust the height of the plane. Zero height is in the middle of the slider track.

Distance

Next to the Height slider bar in the "Cutplane Options" window is a slider bar labeled "Dist." Next to the slider are boxes labeled "Node Vect.", "Nodes", "Node Nos.", "Cell Vect.", "Cells", and "Cell Nos.". When the slider bar is at the bottom, there are two infinite planes parallel to and on the surface of the cutplane, one on each side. As the slider is dragged up, these planes move away from the cutplane in opposite directions while still remaining parallel to the cutplane. If a node or a cell center lies between the two infinite planes, its corresponding point, vector, or number is displayed. This is the basis behind the cutplane distance function. GMV will display nodes, node numbers, node vectors (if any), cells, cell numbers, cell vectors (if any), or all as the distance is adjusted, depending on which option boxes are highlighted.

Adding a cutplane to the main viewer (the manual way)

A cutplane is created by specifying the three noncollinear points needed to define a plane somewhere in space. Choose the "Cutplane" option from the "Calculate" main menu bar and a menu similar to Fig. 4−4 will appear. This menu allows the selection of cutplane creation using a node or cell field, the selection of the desired field in each case, and the creation of multiple cutplanes. To create a cutplane, select an unused number from the lower section of the menu (in this case, "1"). This action will create a menu similar to Fig. 4−5 to allow the definition of the desired cutplane.

Somewhat to the left of this menu and in the center are nine boxes, three rows of three each. Each row defines a point. Enter numbers in the boxes by clicking on the appropriate box and typing in the number. Pressing the tab key will highlight the next box in sequence so that all of the boxes will eventually be filled with numbers. Each row is a point, not each column. When you are satisfied with the points defined, you must choose how to color the cutplane. You can choose either node or cell field values in the first menu (Fig. 4−4). If you choose "Node Field" then each cell will be colored according to its nodes and node field values. If you choose "Cell Field" then each cell will be colored one solid color according to its cell field value. When you have made your choice, click on the "Add" button below the data boxes. If the plane defined by the given points intersects the mesh in any place, GMV will draw a plane there. To remove the plane from the screen, click on the "Delete" button next to the "Add" button. The cutplane will be color coded according to the current node field or cell field, which is displayed on the right side of the window. In Fig. 4−4, the field "pressure" happens to be chosen. This is how color contour plots of data along cutplanes are generated. To change fields, click on the "New Field" button to pop up either the Node Field Selection menu or the Cell Field Selection menu. The "Apply Field Change" button updates the existing cutplane(s) to display the new field information. If multiple cutplanes have been defined, this operation applies the new field to all cutplanes.

Cutplanes may also be temporarily disabled on an individual basis by choosing the particular cutplane from the menu shown in Fig. 4−4, then toggling the "On" button to make the cutplane invisible.

Adding a cutplane the easy way

There is a much easier method of adding a cutplane that does not require you to specify three points. This method is called the "2 Points" method, and its toggle button is located in the upper left corner of the window. Using this function, a cutplane is defined by clicking the mouse on two points on the screen. The cutplane created by this action goes through the two points and is always

normal to the screen.

To define points, first click on the toggle button and move the mouse to the main viewer. The mouse arrow will change into crosshairs. Click with the left mouse button on any two points on the screen. After doing so, GMV will copy these points into the boxes in the cutplane window as well as fill in the missing coordinates needed to make the plane normal to the screen. When the points have been defined, click on "Add" to display the cutplane.

Cutspheres:

Cutspheres are like cutplanes except that data is interpolated where the sphere intersects cell edges. As in cutplanes, there are five cutspheres available, and the display options are the same as for cutplanes. The menus for cutspheres are similar to those for cutplanes, the only difference is in specifying the cutsphere. Create a cutsphere by entering the coordinates of the sphere center and specifying a radius.

Distance:

The "Distance" function in the "Calculate" menu is used for calculating the linear distance between two points along the screen. To use the distance function, first choose it from the menu. As soon as the function is chosen, the mouse pointer will change into crosshairs resembling a plus sign when it is in the main viewer. Click the crosshairs on one of the two points between which the distance is to be calculated. A small grey dot will appear. Move the mouse to the other point and click the crosshairs there. After both points have been defined, GMV will display the distance in the upper portion of the main GMV window to the right of the background color controls. Each time a distance is calculated, the old number next to the color controls will be overwritten with the new one.

Field Calc.

The "Field Calc." option of the "Calculate" menu is used to calculate data from an existing node field or cell field using supplied unary or binary operations. The calculated data is placed in one of the five extra fields provided.

Selecting a field to build

Choose the "Field Calc." option from the "Calculate" main menu bar then select either "Node Field" or "Cell Field" (if available) and a menu similar to Fig. 4.7 will appear. The "FldCalc" names indicate that data for the field has not been calculated. Select one of the five fields and a menu similar to Fig. 4−8 appears.

Build (calculate) the new field

To build the new field, first select an operator from the

Operators list. Note that the operators refer to the variables X, Y, and C. The X and Y variables refer to selected fields, while the C variable represents a user defined constant. Next, fill in the constant field "C=", if needed, and check that the current fields displayed the "X Field" and "Y Field" are the fields to operate on. If

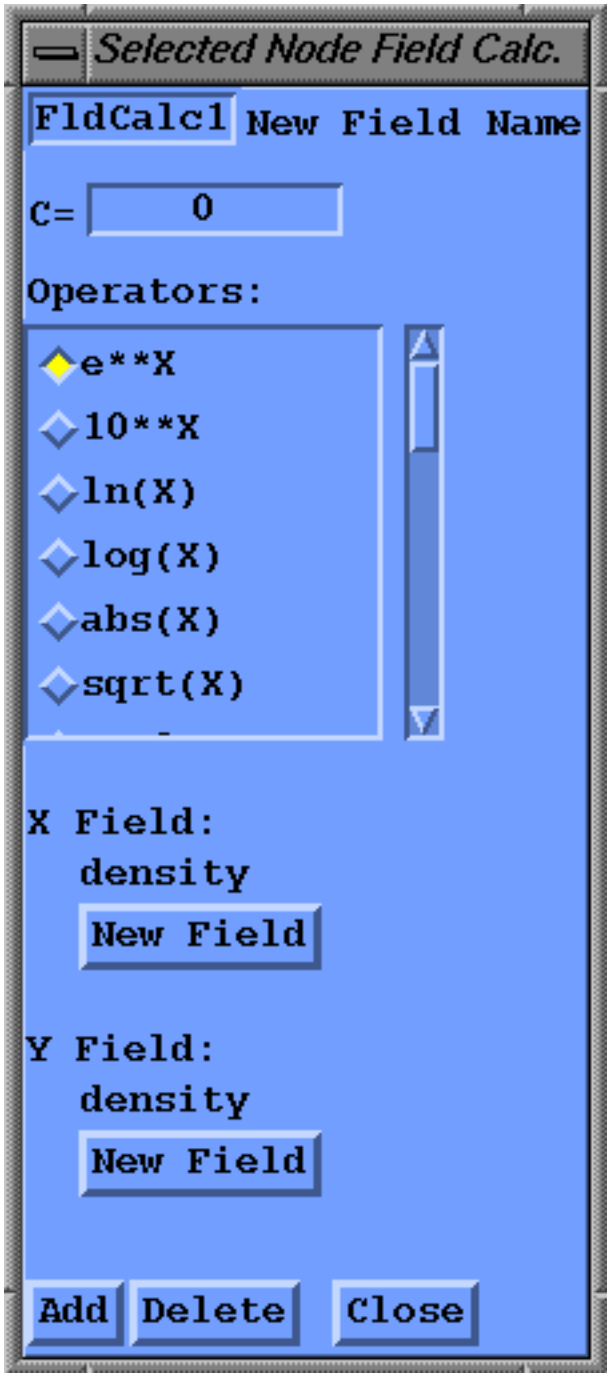


Figure 4−8. Field Calc. Build menu

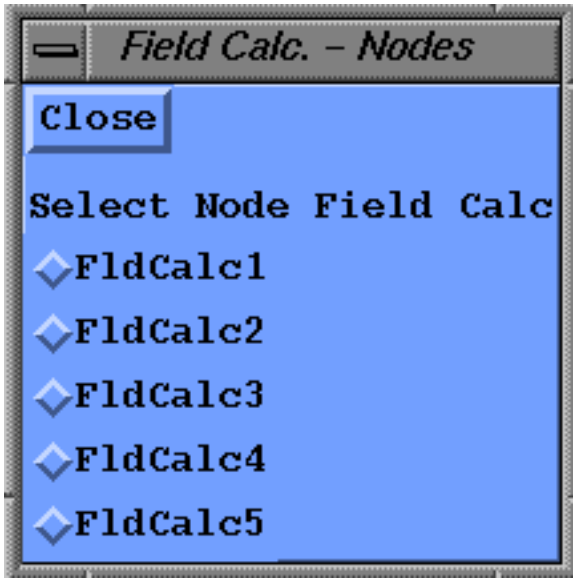


Figure 4−7. Field Calc. Selection menu

not, click on the appropriate "New Field" button to select a field from the Field Selection menu. Finally, enter a name in the "New Field Name" text field and click on the "Add" button. If an invalid operation, such as the square root of a negative number, is recognized for a field data element, a warning message will appear on the menu and that data element will not be changed. The new field is now available for use as is any other field and is identified by the new field name. Use the "Delete" button to delete the selected calculated field. When a calculated field is deleted, the field name changes to the "FldCalc" prefix.

Grid Analysis:

"Grid Analysis" allows you to selectively view only a portion of the mesh data contained in a GMV input file. This is done by specifying individual nodes or cells to be viewed on the screen. This function is especially useful when there are

thousands of nodes and cells in a given set of mesh data. The ability to isolate individual areas of data allows for more detailed analysis of the problem at hand. To open the grid analysis window, use the mouse and choose "Grid Analysis" from the "Calculate" menu. A menu similar to Fig. 4−9 will appear.

Selecting cells by nodes or cell numbers

Cells can be selected by cell numbers or by specifying nodes that are common to one or more cells. In the window are two columns of eight boxes each used for data entry. The first column is labeled "Select cells by nodes." Here you can give GMV the numbers of a few of the nodes. Based on this information, GMV will decide which cells are relevant and display them on the screen. GMV will display the cell that contains all of the given nodes as vertices. Nodes that are not vertices of any cell are ignored by GMV. If two or more cells contain the same number of the given nodes, GMV will display all of the relevant cells. The second column of data boxes is labeled "Select cells by number." Here you may list the numbers of the cells to view. Fill as many boxes as necessary with the relevant cell numbers (there is a maximum of eight boxes). To view a specific range of cells, such as cells numbered from 1 to 50, choose cells by placing the numbers of the first and last cells of the series in the boxes at the bottom

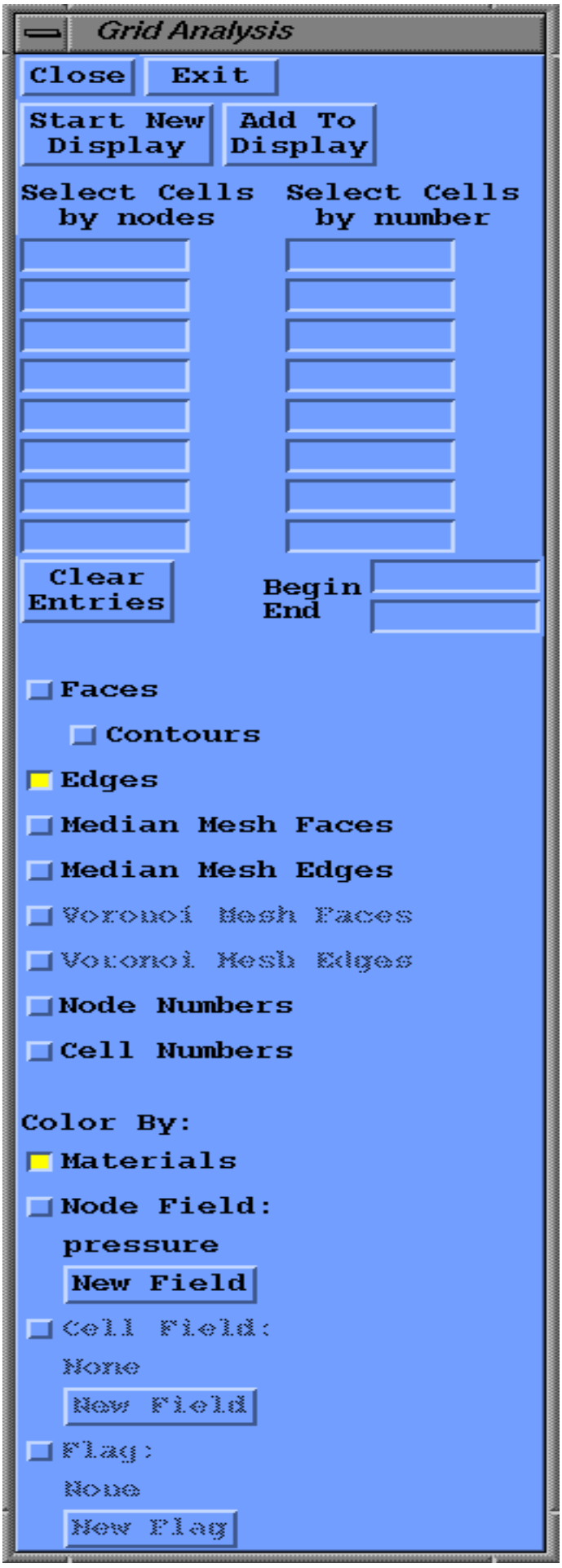


Figure 4−9. Grid Analysis menu

of the submenu. These boxes are labeled "Begin" and "End." GMV will show all of the cells in that particular series.

After all of the node and cell selection data has been entered into the appropriate boxes, direct GMV to display the requested cells. There are two buttons at the top of the menu that accomplish this. They are labeled "Start New Display" and "Add to Display." If the "Start New Display" button is used, GMV will erase any objects the grid analysis function created beforehand, before it draws in the selected cells. If the "Add to Display" button is used, GMV will add the newly chosen cells to the display. The cell drawing commands in the "Grid Analysis" menu are the same as those for the "Cells" display menu with the addition of "Node Numbers". These drawing commands only apply to nodes and cells selected by grid analysis. All other drawing options are still available.

Note that the "Grid Analysis" menu is created with the "Edges" button selected as the default. When nodes or cells are specified, the default behavior is to draw the cell edges in the display. The node numbers of the cells of interest are only displayed when the "Edges" button is selected. As such, if node numbering is desired but does not appear as expected, ensure that the "Edges" button is activated.

To exit "grid analysis", click on the button labeled "Exit" in the upper left corner of the menu. This action deletes the grid analysis cell drawing and closes the menu.

Color By:

Cell faces and edges are colored the same way as in the "Cells" menu under the "Display " option.

Median and Voronoi mesh.

The Median mesh options generate the median mesh edges and faces as selected. The Voronoi mesh option generates voronoi mesh edges and faces for tetrahedaral meshes only.

Isosurfaces:

An isosurface is the surface in space where a selected field is always a particular value. For example, the isosurface where the voltage around a charged particle is constant would be a spherical shell. Isosurfaces for any field in a GMV input file can be generated using this tool, (see Figs. 4−9 and 4−11).

Adding a material isosurface

GMV can generate an isosurface from node material data. Because of the noncontinuous nature of material data, there is a special method for drawing these surfaces. Even if the material data is cell centered, each node is assigned a

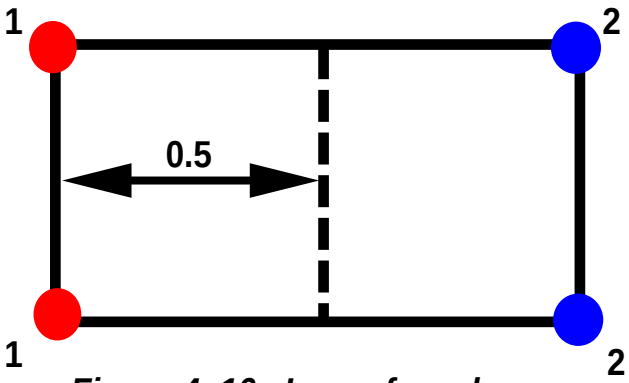


Figure 4−10. Isosurface demo.

material number. Click on "Calculate" from the main menu bar. Click on "Isosurfaces." A submenu with two options will appear to the right. Choose "Materials" to bring up the "Material Isosurfaces" menu, (see Fig. 4−11). To add a material isosurface, first adjust the material fraction ("Mat. Frac.") slider bar above the list of materials. The material fraction tells GMV how far between differing materials to draw the surface. For example, take a rectangle with four nodes. Two of the nodes are material number 1, and the others are material number 2. With a material fraction of 0.5, GMV would draw the isosurface halfway in between the two different materials, (the dashed line in Fig. 4−10). After the material fraction has been adjusted, click on "Add" next to the box corresponding to the desired material to add the isosurface. Click on "Del" to remove the surface. Note: truer material surfaces can probably be generated by simulation code and entered as surface polygons.

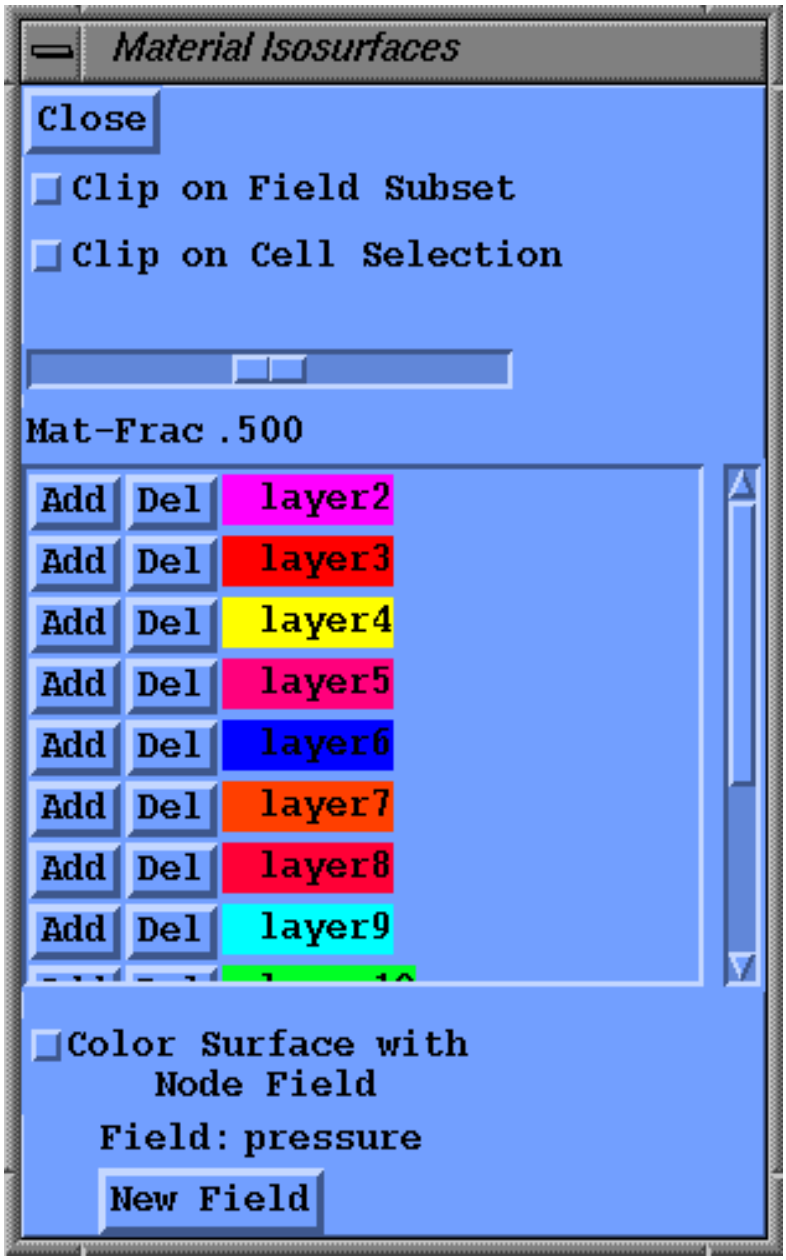


Figure 4−11. Material Isosurface menu

Adding a field isosurface

Open the "Field Isosurface" menu from the main menu bar under "Isosurfaces" in the "Calculate" menu. A

menu similar to Fig. 4−12 will appear. To add a field isosurface, first check that the desired node fields is the current field. The current node field is displayed after "Field:" near the bottom of the "Field Isosurfaces" menu. To change the current field, click on the "New Field" button to pop up the Node Field Selection menu Below the current field is a data entry box into which a contour value must be entered. The "Contour Value" is the number in the selected field that will generate the surface with that value. To enter a number, click on the box and enter a value between the minimum and maximum values for the selected field. The range of min. and max. values appears on the right− and left−hand side of this particular box. Finally, you must choose a color for the isosurface. Down the middle of the "Field Isosurface" menu is a column of colored boxes numbered one through twenty. Choose a color for the isosurface. Click on "Add" next to the desired color. GMV will beep and display the isosurface. If the contour value is not within the range specified on the sides of the contour value data box or if the isosurface for a particular value simply does not exist, GMV will show nothing. GMV lists the field and contour value of each isosurface next to the surface's color box in the isosurface window. To delete an isosurface, click on "Del" next to the appropriate isosurface color box. Isosurfaces can be turned on and off

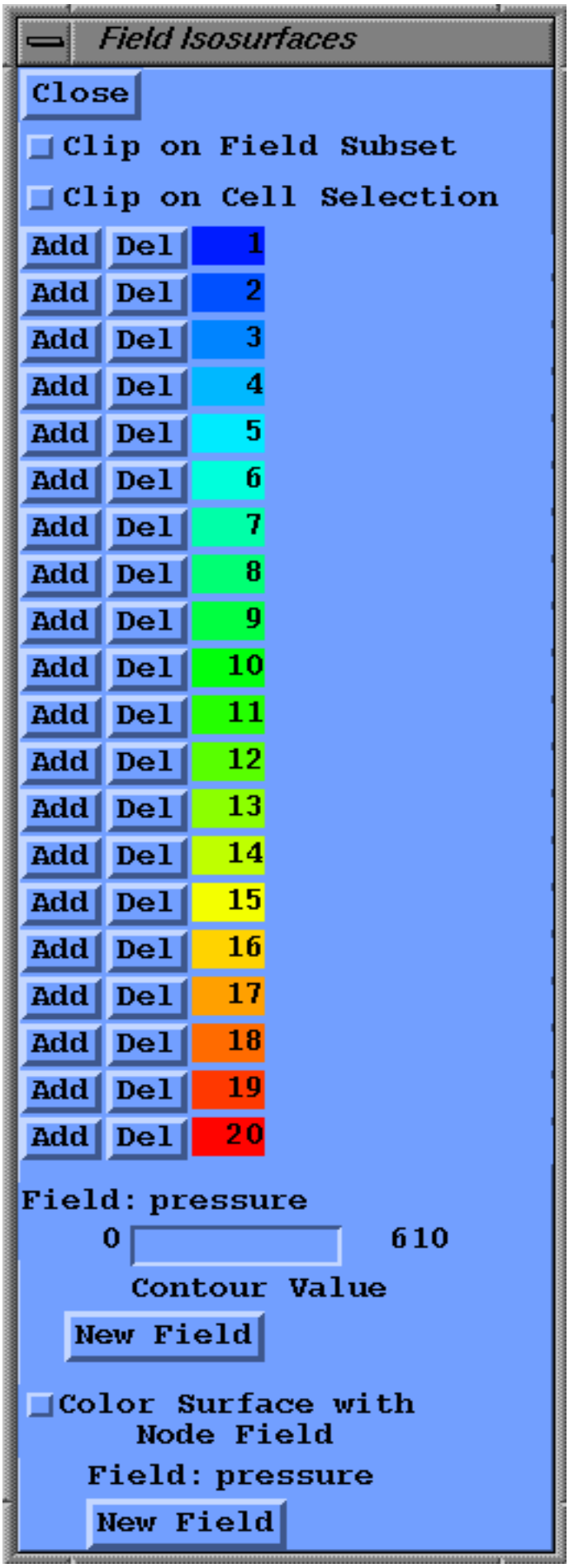


Figure 4−12. Field Isosurface menu

by clicking on the colored box.

Clip on field subset and cell selection

These two toggle buttons tell GMV to draw isosurfaces based only on data contained within some subset. For the "Clip on Field Subset" option, the subset is defined by the "Subset" function. The "Subset" function is found in the Controls−2 menu. This function is explained later on in this text. For the "Clip on Cell Selection" option, the subset is defined by the Boolean expression created using the "Select" option in the "Cells" menu. Isosurfaces will then only be drawn from node data found in the selected cells.

GMV only checks the status of these two buttons when an isosurface is created. Therefore, any changes made with these two options will not take effect until a new isosurface is calculated.

Figure 4−13. Isovolume menu

Coloring isosurfaces with field values

The isosurfaces can be colored with the blue−to−red intensity color of node field values. All the isosurfaces for a specific type (material isosurfaces or field isosurfaces) are colored according to the selected field values. To color the isosurfaces with field values, first select a field with the "New Field" button, then click on the "Color Surface with Node Field" button.

Isovolume:

The "Isovolume" option of the "Calculate" menu allows the creation of isovolumes (the display of a volume with surfaces of which are interpolated between the selected minimum and maximum field values). The selection of this option creates the menu shown in Fig. 4−13.

This menu lists the current active field, and the minimum and maximum values of this field. To add an isovolume, enter the value for the minimum and maximum surfaces in the "Isovolume Range" areas, then select

"Add." The isovolume may be toggled on and off using the "On" button. The "Clip on Field and Cell Selection" functions behave similarly to the same buttons described for the "Isosurfaces" menu. The isovolume surface can also be colored using node field values as described for isosurfaces.

Query Data:

The "Query Data" option of the "Calculate" menu makes it easy to find information about any cell or node in the grid. Query Data also has facilities to interactively probe the image for a node or cell number. There is a function to obtain node numbers by node field value.

Getting node and cell values

Choose the "Query Data" option from the "Calculate" main menu bar and a menu similar to Fig. 4−14 will appear. The menu has two main parts. The left side displays information about a selected node and the right side has information about a selected cell. To list all the data associated with a specific node, enter the number of the node in question in the box labeled "Node:" and click on the "Get Values" button. All of the available information about the specified node will appear below. The first three numbers are the node's X, Y, and Z coordinates. The three numbers after that are the magnitudes of the i, j, and k velocity vectors if any exist, which are labeled U, V, and W respectively. After the velocity, GMV will list all field values for the node (i.e. temperature, pressure, or speed data). Finally, GMV lists the node material and flag values. Below the node information box, the distance between the last two selected nodes is listed.

The "Cell Info" side of the window can be used to retrieve information about individual cells. To list the data associated with a specific cell, enter the number of the desired cell into the box labeled "Cell:" and click on the button labeled "Cell Values." All of the available data for that cell will be displayed below the box with the cell number in it. First is the cell center's X, Y, and Z coordinates followed by any cell centered field data, then the cell material and flag values followed by the numbers of all the nodes that make up the vertices of the cell. Note that any cell−centered data is also displayed as a node value. GMV averages all cell−centered data to the nodes. Any data defined to be cell−centered in the input file will be averaged to the nodes and be displayed on the node value side of the grid value window.

Probing node and cell numbers from the image

The "Get Node Data From Screen" button allows you to use the mouse to get node data from a node displayed on the screen. This feature is available whenever nodes are displayed by the "Nodes" display option. Click on the "Get Node Data From Screen" button and move the cursor to the image, then move the crosshair

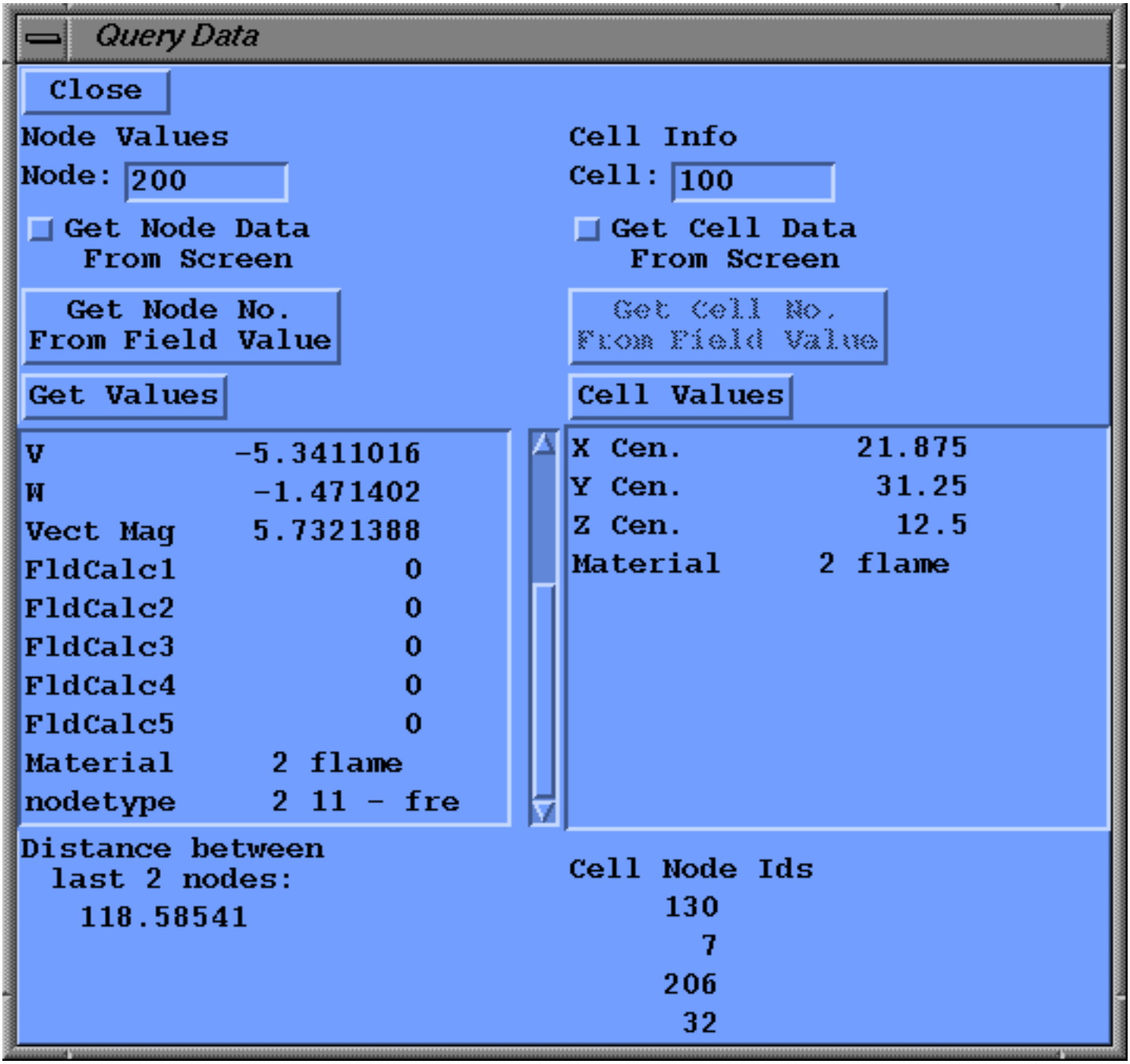


Figure 4−14. Query Data menu

cursor to a node and press the left mouse button. The number of the node closest to the crosshair will be displayed in the "Nodes:" box. Also, the data for the node will be updated. You may continue to probe nodes until you click off the "Get Node Data From Screen" button . Similarly, the "Get Cell Data From Screen" button returns the cell number and data of the cell whose cell center is closest to the crosshair when cells are displayed.

Getting node and cell numbers by field value

You can retrieve a node number by node field value by clicking the "Get Node No. From Field Value" button. When selected, a menu similar to Fig. 4−15

appears. Select a field to query and the field data minimum and maximum are displayed along with the first node number at these extremes. If there is more than one node with a minimum or maximum value, the number of matches is displayed. To get the node number closest to a specific value for the current field, enter the value in the text box labeled "Match Value" and click on the "Get Node" button. The node number closest to the match value is displayed along with the exact value of the node. If there is more than one node that has the exact match value, the number of matches is displayed instead of the node value. Similarly, use the "Get Cell No. From Field Value" button for cell−centered field values (if any).

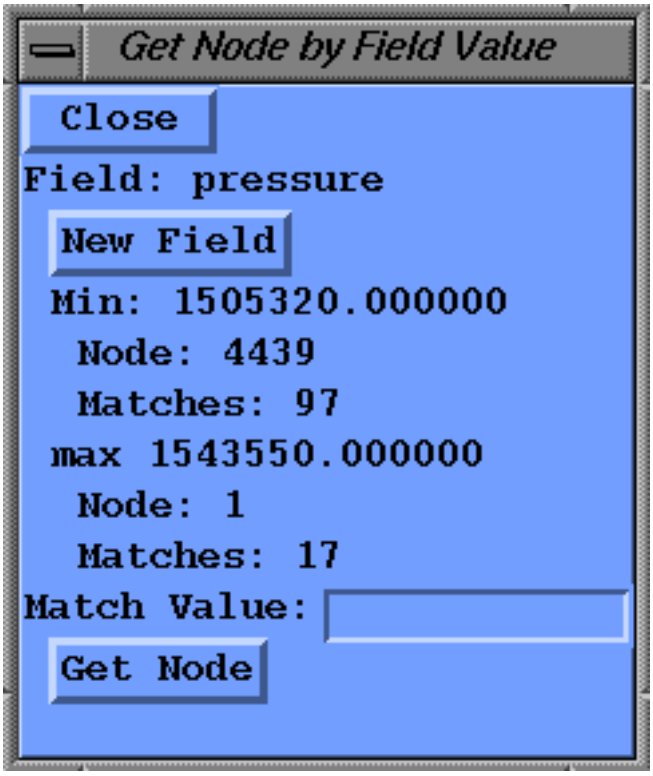


Figure 4−15. Get Node by Field Value menu

The Ctl−1 (Controls−1) Menu

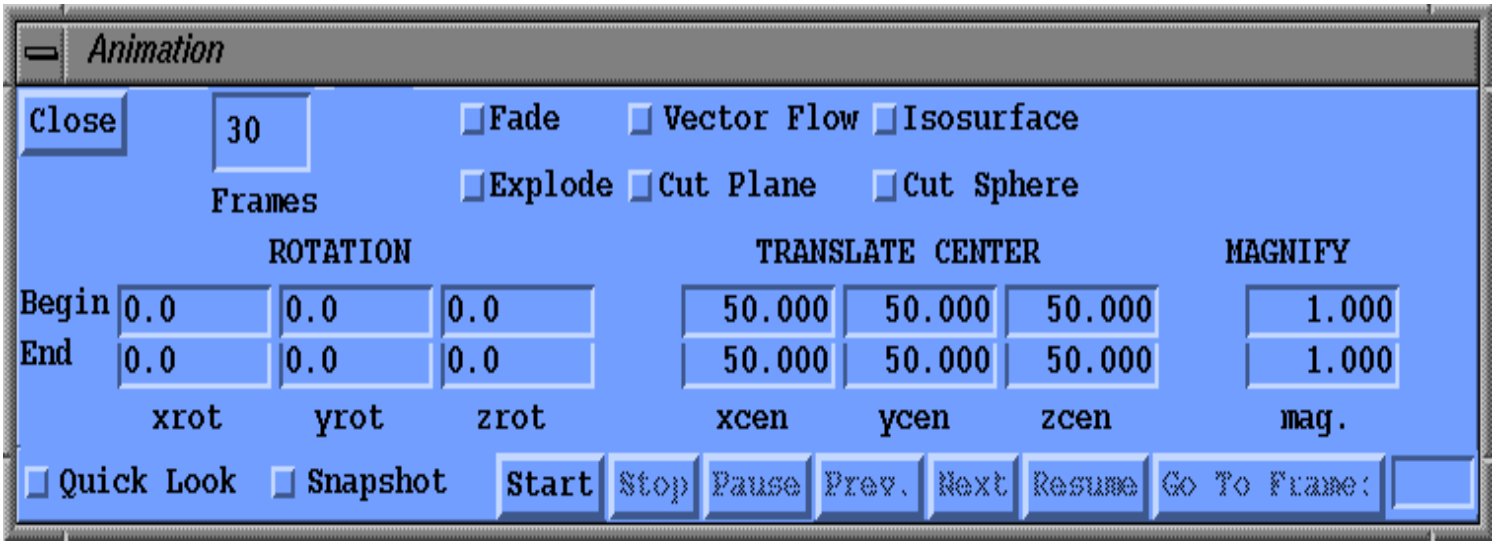


Figure 5−1. Animation menu

Animation (orthographic and perspective modes)**:**

GMV can be used to create animation sequences. To open the animation menu, choose the "Animation" option from the "Controls−1" main menu bar. There are two different animation menus, depending on which "View" mode you are currently in. If you are in orthographic or perspective modes, a menu similar to Fig. 5−1 should appear. To begin an animation, click on the "Start" button. Click on "Stop" to stop the current sequence. A given animation sequence may be paused at any time during its run by clicking on the "Pause" button. When paused, the animation sequence may be stepped through one frame at a time by clicking on the "Prev." and "Next" buttons located at the bottom of the menu. You may jump to a specific frame by entering its number in the box with the "Go To Frame" button next to it and clicking on that button. To resume the animation where you left off, click on the "Resume" button.

Number of animation frames

In the upper left corner of the "Animation " menu is a box labeled "Frames." Enter the number of frames the sequence should contain. Thirty is the default (approximately one second of animation on video). The more frames there are, the smoother the animation sequence.

Rotation

The object in the main viewer can be rotated during the animation sequence by specifying an angular distance in degrees for the object to move. In the left center of the animation window are six boxes with the heading "Rotation." The top row of three boxes is labeled "Begin" and the bottom row of boxes is labeled "End." By specifying beginning and ending angles in the appropriate boxes, the object can be rotated about any axis relative to the current view angles. For example, if you want to begin at 90 degrees of rotation and end at 180 degrees, then enter 90 in the top and 180 in the bottom of the X−rotation column. This would rotate the object in the main viewer about an axis going through the center of the object parallel to the X−axis, exactly 90 degrees. The same is true for the Y and Z rotation columns. X, Y, and Z rotations can be done simultaneously by entering the appropriate numbers in the boxes, but this process is not recommended. If no rotation is desired, enter the same number (usually zero) in both the begin and the end boxes to achieve this effect.

Center translation

The object in the main viewer can be moved linearly in any direction during an animation. This motion is accomplished by entering data into the set of six boxes with the header "Translate Center." The first row of three boxes is used to tell GMV where to start the center translation in the form of X, Y, and Z coordinates. The second row tells GMV where to end the movement. GMV takes default values from the "Center" tool also found in the "Controls−1" menu.

Magnification

The magnification of the object in the main viewer can be increased or reduced during an animation sequence. On the far right of the animation window are two boxes with the header "Magnify." In the top box enter the starting magnification and in the bottom box enter the ending magnification. During the sequence, the object will either grow larger or smaller depending on whether the beginning magnification is greater than or less than the ending. If no change in magnification is desired, enter the same number into both boxes (usually 1.00). The object will then remain the specified size throughout the entire animation sequence.

Vector flow

The vector flow option displays the paths that particles with vector data would take. GMV shows the vector flow by dividing the length of the each node’s current vector into as many equal pieces as there are frames and then displaying each piece in sequence during the animation. To include vector flow in an animation, turn vectors on and then click the toggle button in the upper right corner of the animation submenu before starting the animation.

Cutplane

The animation window includes provisions for sweeping a cutplane across the object in the main viewer. The cutplane may sweep perpendicular to the X, Y, or Z axes. Only one sweeping cutplane may be in an animation at a time. To insert this effect into the animation, click on the box with the label "Cut Plane" in the upper right corner of the animation submenu. A submenu similar to Fig. 5−2 should appear. The first four options allow choosing along which axis the cutplane should sweep. GMV can only sweep across one axis at a time. Clicking on the "None" option will turn off the cutplane option. Below the direction buttons is the "Options" button. Clicking on this button displays the "Cutplane Options" menus (see Cutplane in the Calculate section) to define drawing options. After the "Options" button are two slider bars. These sliders control where the cutplane sweep begins and ends along the chosen axis. The default is to sweep the entire length of the object. During an animation sequence that includes a sweeping cutplane, the coordinate on the selected axis where the current cutplane lies will be displayed at the bottom of "Cut Plane" menu. Below the node and cell field label is the currently selected field for interpolation onto the cutplanes. Click on the "New Field" button to select a new field from the Field Selection menu. When all the selections have been made, click on "Close" to remove the submenu from view.

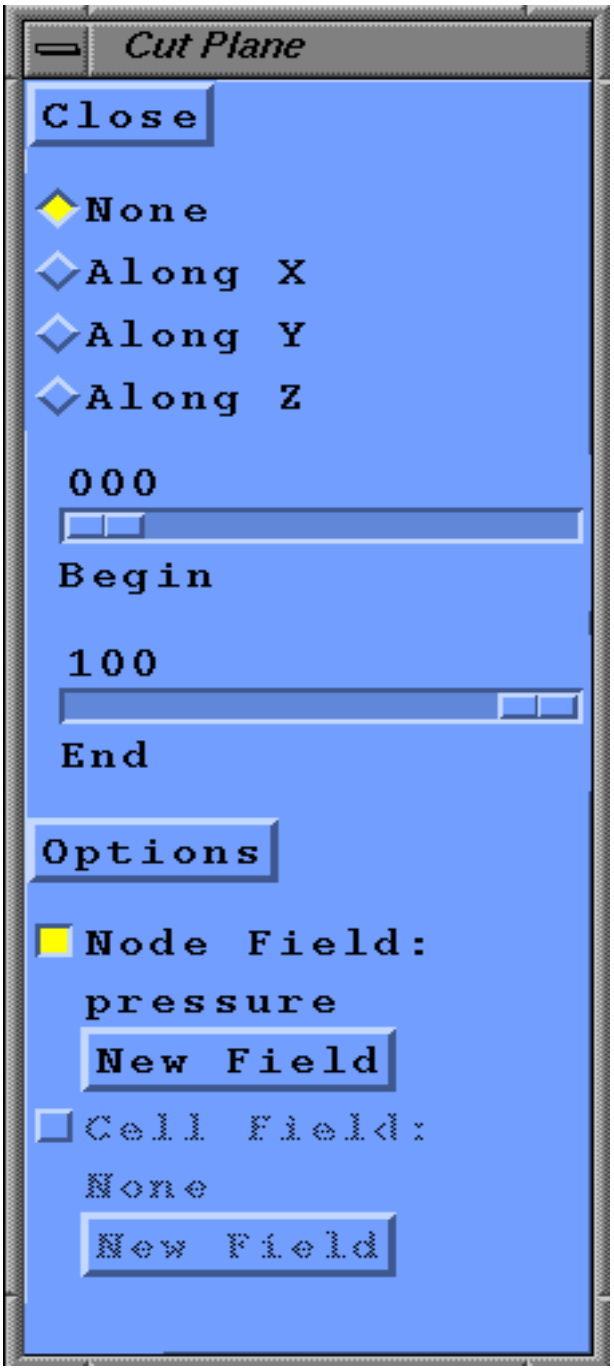


Figure 5−2. Cutplane Animation submenu

Fade

The fade option allows for fading out polygons and isosurfaces over the course of an animation sequence. To use the fade option during an animation, click on the toggle button labeled "Fade" in the animation menu. A submenu similar to Fig. 5−3 will appear. First, set the beginning and ending transparency factors

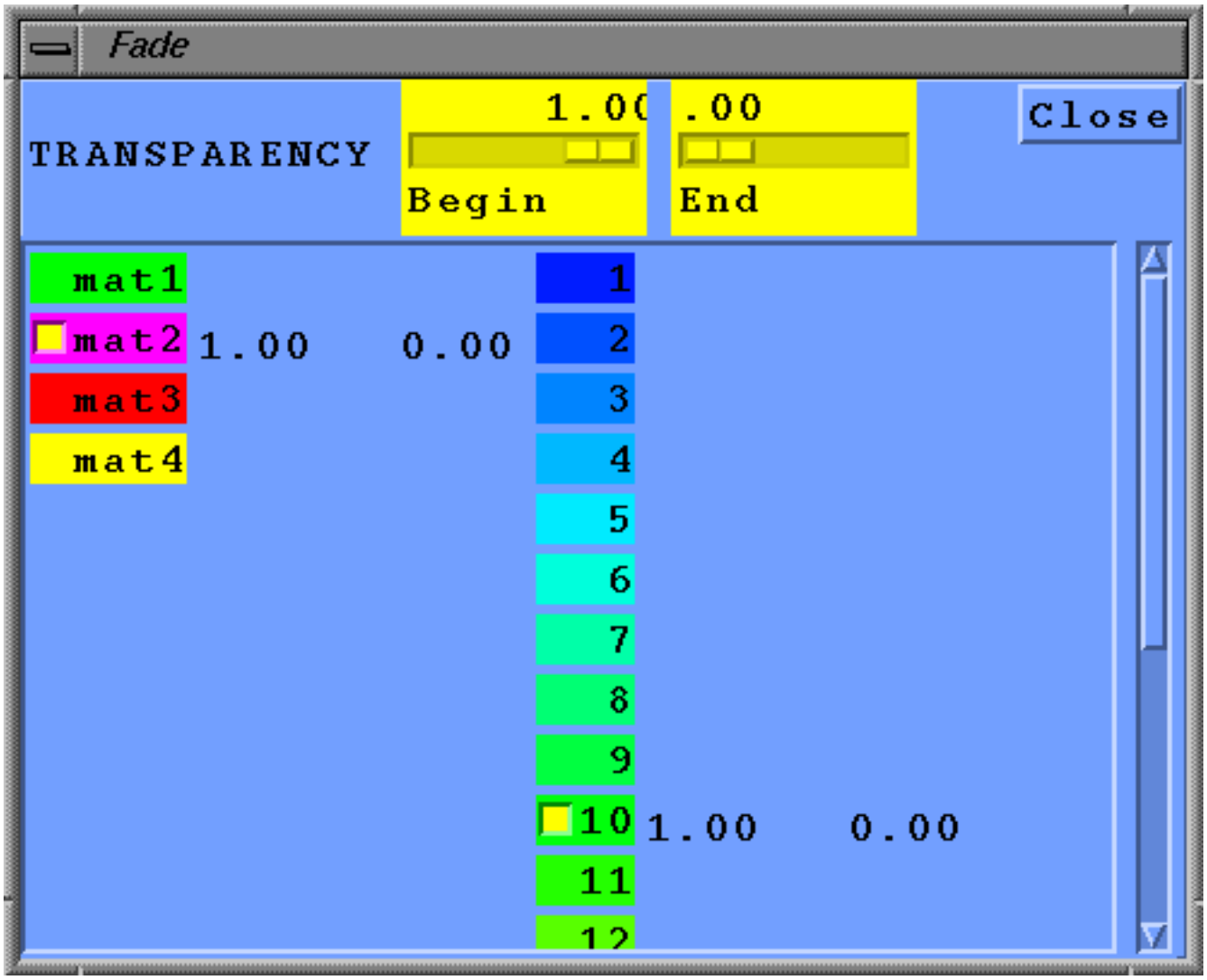


Figure 5−3. Fade Animation submenu

using the yellow slider bars at the top of the submenu. A value of one corresponds to no transparency and a value of zero is 100% transparent. Next, choose the material or isosurface to be faded during the animation by clicking on the appropriate box. When a selection is made, the beginning and ending transparency factors will appear to the right of the box. When finished choosing things to fade, click on "Close" to remove the submenu from view. The toggle button in the animation submenu will remain on, indicating that fading will be included in the next animation. To turn off "Fade," toggle all of the material and isosurfaces in the fade submenu off. Now, when the window is closed, the fade option will also be toggled off.

Exploding cells or polygons during animation

You can explode polygons and cells during an animation sequence. Toggle the button labeled "Explode" in the animation menu. A submenu similar to

Fig. 5−4 will appear. With the two slider bars at the top, adjust the beginning and ending explode percentages. Next choose polygons and/or cells to explode. The beginning and ending explode percentages will appear to the right of whatever was chosen to explode during the animation. For cell explode, choose material or the flag to separate the cells. Click on "Apply" to activate your choices. After choosing the various options, click on "Close" to remove the submenu from view. The "Explode" toggle button in the animation menu will remain on until both "Polygons" and "Cells" are toggled off in the explode window.

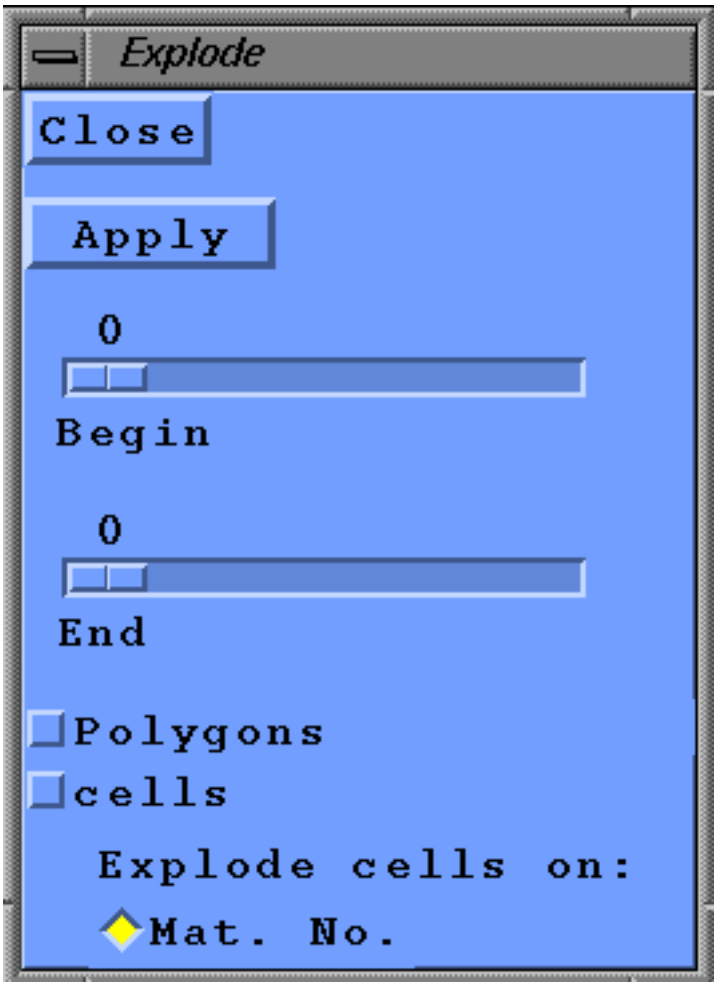


Figure 5−4. Explode Animation submenu

Snapshot

Each frame of an animation sequence can automatically be saved using the snapshot function in the animation menu. Highlight the "Snapshot" option to the left of the "Start" button. When this box is highlighted, GMV will save each frame of the animation sequence in its own file. All of the files from the same animation will have the same prefix. When the "Start" button is pressed, a File Selection Menu will appear; enter the file prefix and click the "OK" button. GMV will append the frame number of that particular snapshot to the file prefix. For example, snapfile007 indicates that the file contains the seventh frame of the sequence. The animation snapshots are created in movie size (720x486) so they can be combined with time sequence images to create movies. Note: the "Pause" button is not active while saving snapshots.

Quick look

The "Quick Look" option tells GMV whether or not to use the current interactivity setting during the animation. The interactivity can be adjusted with the slider bar on the right of the main GMV window labeled "Int." Without "Quick Look" selected, the animation will be carried out with all objects

drawn. This option can speed things up, especially when many objects are drawn.

Isosurface animation

Isosurfaces may be added to an animation sequence by toggling the "Isosurface" button in the "Animation" menu. A submenu similar to Fig. 5−5 will appear. The first two radio buttons turn the isosurface animation on or off. Click "On." Underneath the on and off buttons is the current field from which isosurfaces can be calculated. Click on the "New Field" button to select another node field from the Node Field Selection menu. After a field is chosen, GMV places the default data range for that particular field in the "Begin" and "End" boxes near the bottom of the window. The default data range is defined as the minimum and maximum values in the current field. To change the data range, click on either the "Begin" or "End" data box and enter the desired value. GMV will divide the data range into as many equal increments as there are frames. During the animation sequence, a new isosurface will be drawn for each contour value in the data range, one for each frame. Also during the animation sequence, the current isosurface contour value will be displayed at the bottom of the "Isosurface" menu. The color of the isosurface animation can be changed using the "Color Edit" function in the "Controls−1" menu. Click on "Close" to remove the submenu from view. The next time an animation sequence is started, isosurface animation will be included. To turn off isosurface animation, reopen the isosurface submenu and click the "Off" button. When the

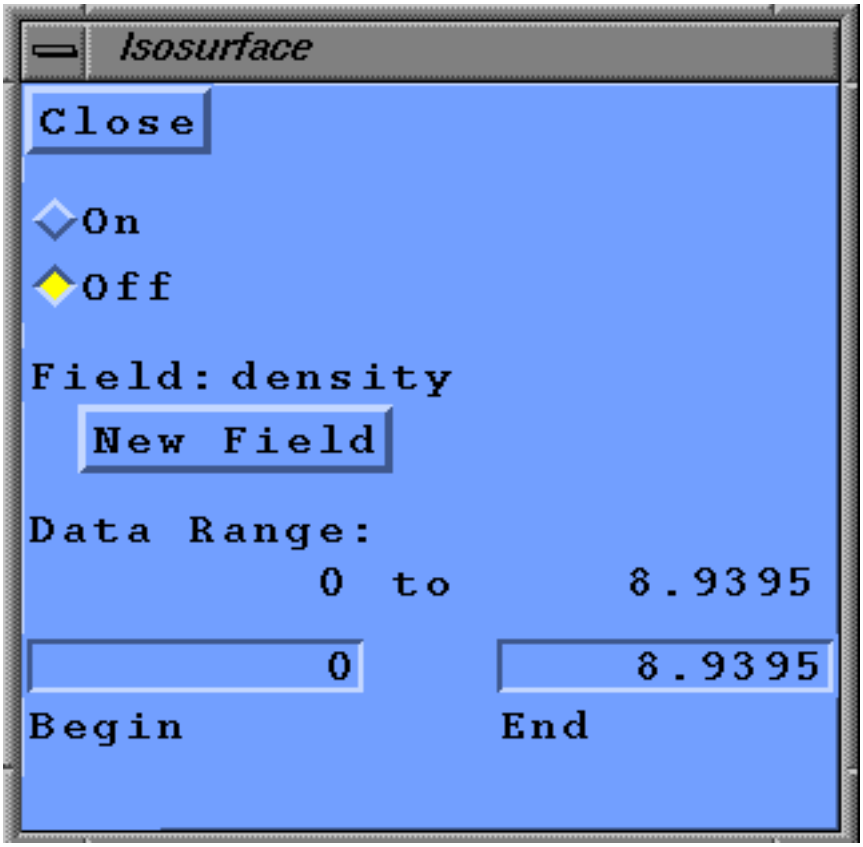


Figure 5−5. Isosurface Animation submenu

window is closed, the isosurface button in the animation menu will not be highlighted.

Cutsphere animation

Cutspheres may be added to an animation sequence by toggling the "Cutsphere" button in the "Animation" menu. A submenu similar to Fig. 5−6 will appear. The first two radio buttons turn the cutsphere animation on or off. Click "On." Underneath the on and off buttons are the input text fields that defines the center of the spheres, and the beginning and ending radii for the spheres. Below the text fields is the "Options" button. Click on this button to set drawing options. After the "Options" button is the current field from which cutspheres can be calculated. Click on the "New Field" button to select another node field from the Node Field Selection menu. During the animation sequence, a new cutsphere will be drawn for each sphere radius between the beginning and ending radius, one for each frame. Also during the animation sequence, the current cutsphere radius will be displayed at the bottom of the "Cutsphere" menu. Click on "Close" to remove the submenu from view. The next time an animation sequence is started, cutsphere animation will be included. To turn off cutsphere animation, reopen the cutsphere submenu and click the "Off" button. When the window is closed, the cutsphere button in the animation menu will not be highlighted.

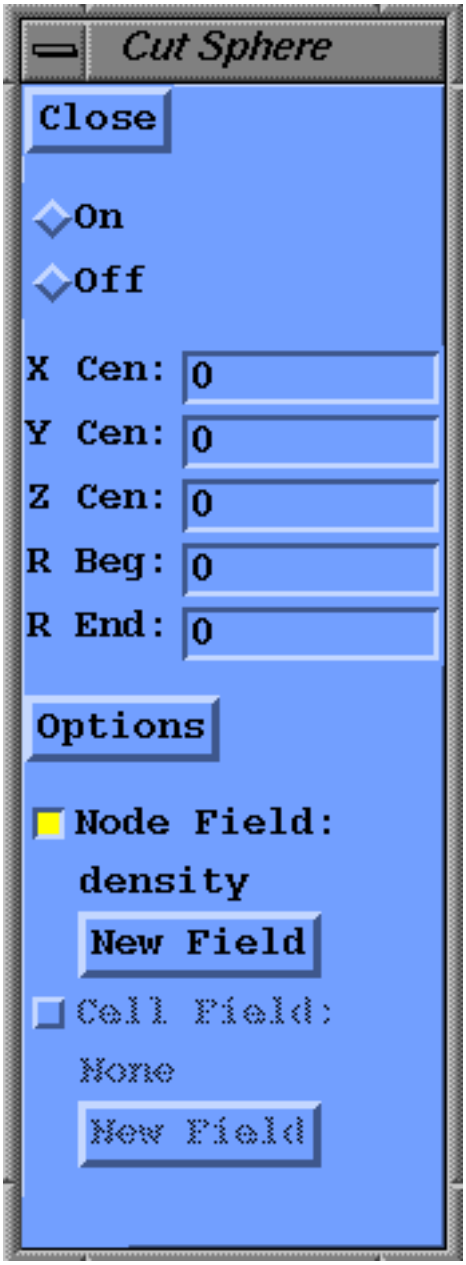


Figure 5−6. Cutsphere Animation submenu

Animation (flight mode):

In flight mode, an animation menu will appear. (See Fig. 5−7) Flight animation moves the simulated helicopter from point to point in space, going up, over, around, and through the object in the main viewer. To start an animation click on "Start." To stop the animation, click on "Stop."

Setting control points

Control points are points in space that define a path for the flight mode simulated helicopter to follow. In order to set control points, one should be familiar with the flight mode mouse controls. For a review of these controls, please refer to the **View Menu** section.

To set a control point, first select the number of frames wanted for each control point. The default is 30, but it can be changed by clicking on the box labeled "Control Point Frames" and entering a new number. Next, move the simulated helicopter to the location of the first control point using the mouse. Click on "Set New Control Point" when the location is reached. The number of frames for that control point will appear next to the control point's number in the list at the bottom of the window.

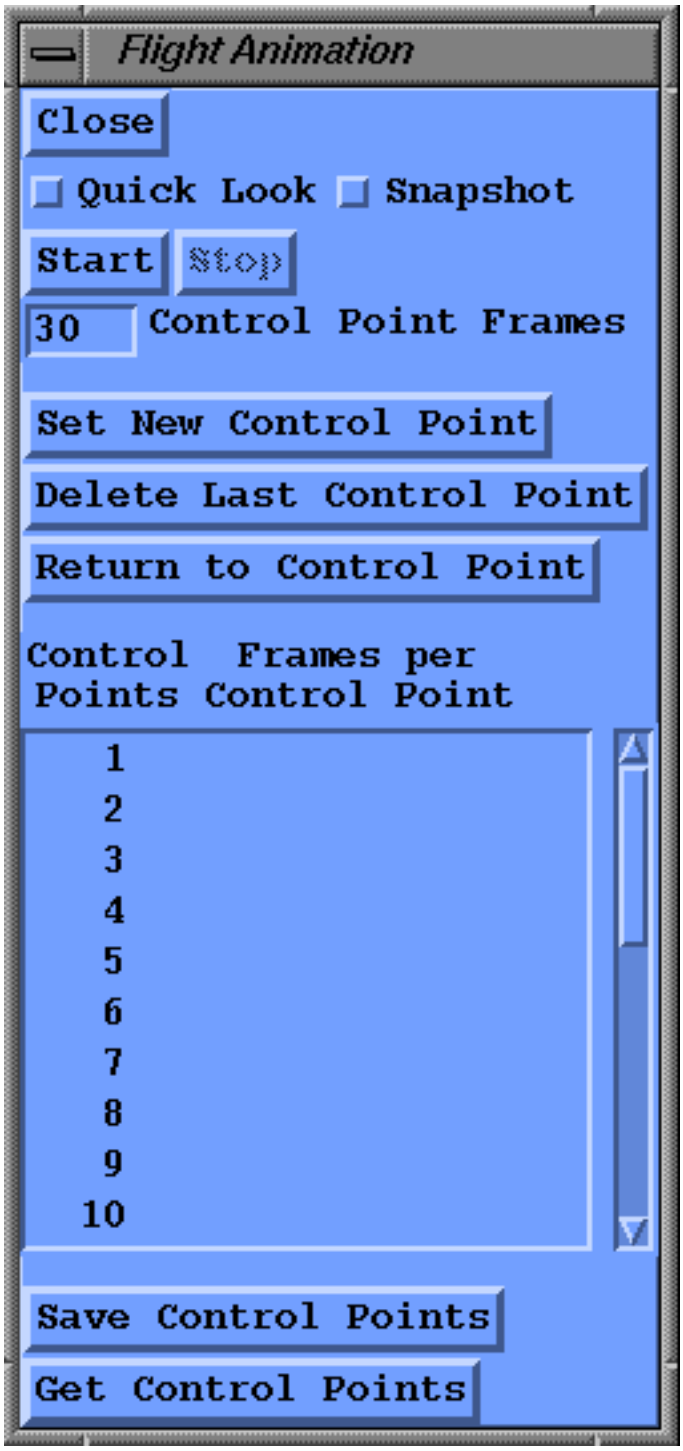


Figure 5−7. Flight Animation menu

When all control points have been set , click on "Start" to begin the animation.

To delete a control point, click on the button labeled "Delete Last Control Point." This will clear the most recently entered control point. Control points must be deleted sequentially from last to first and cannot selectively be removed by number.

Should a mistake in entering a control point be made, there is a way to return to the most recently set control point so that, by retracing steps, one can try again. To go back to the last control point, click on "Return to Control Point."

Saving and Retrieving control points

At the bottom of the "Flight Animation" menu are two buttons. These two buttons, labeled "Save Control Points" and "Get Control Points" allow you to save and retrieve control points. A file Selection Menu will appear to to select or enter the file name to save or retrieve.

Quick look and Snapshot

If the object in the main viewer is large and complicated, you can choose to incorporate the current interactivity setting into the animation. Click on the "Quick Look" toggle button.

GMV can also take snapshots of every frame during the animation sequence. To do this, click on the "Snapshot" toggle button. When the "Start" button is clicked, a File Selection Menu will appear to enter the file prefix. Each snapshot file will then begin with the same prefix followed by a three digit number indicating the frame number of that particular snapshot. All snapshots are in Silicon Graphics RGB format.

Axes:

"Axes" is the second option in the "Controls−1" menu. Its function is to turn on or off the X, Y, and Z axes in the main viewer. To activate this function, choose it from the menu. The axes will appear or disappear depending on whether or not the axes are currently showing. Turning off the axes in the main viewer will have no effect on the axes orientation view box located in the upper right corner.

Beep Sound:

"Beep Sound" toggles the GMV beep sound on/off.

Bounding Box:

The bounding box is a six−faced volume that encloses data in the main viewer. It can be turned on or off. By default, the box is set to be the smallest volume that will enclose all of the data in the main viewer. However, the size of the bounding box can be changed using the "Bounding Box" option. A menu similar to

Fig. 5−8 will appear. To turn on the bounding box, click on the box labeled "On" in the lower−left corner. Next to the "On" button for the bounding box is the "Show Coordinates" button. This toggle button allows you to turn the coordinates on or off depending on your preference. The length, width, and height of the bounding box are colored red, green, and blue, respectively. Next to the point where the three colored lines meet are the coordinates for this point, which is the origin of the bounding box. There is only one number on the opposite ends of the colored lines.

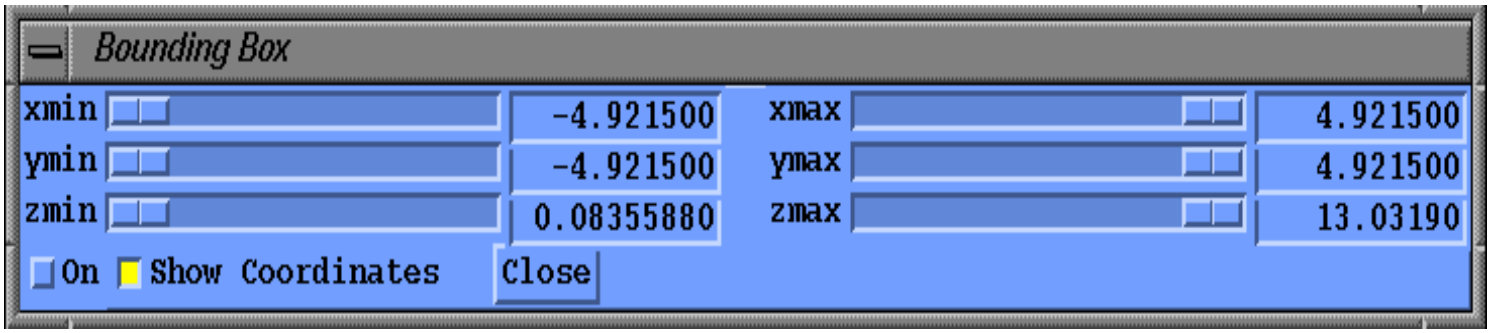


Figure 5−8. Bounding Box menu

Only the coordinate that is different from the origin coordinates is listed here. For example, on the far end of the red line, only the X coordinate is listed because it is the only one that is different from the origin coordinates. With the numbers provided, it is easy to determine the size of the object in the main viewer. The size of the box is changed by adjusting the six sliders in the menu or by entering the numbers directly. There is a slider bar for the minimum and maximum values for each axis. As you slide the bar back and forth, you will see the box change size. The bounding box cannot grow any larger than the 3−D plotting box. When finished adjusting the bounding box, click on "Close" to shut the menu.

Center:

The centering tool is used to tell GMV where the center of the current object lies. The center is important because it determines how GMV rotates the object in the main viewer. The values found in this tool also affect any center translation done during an animation. To open the "Center" window, choose the option from the menu. A submenu similar to Fig. 5−9 appears. There are three

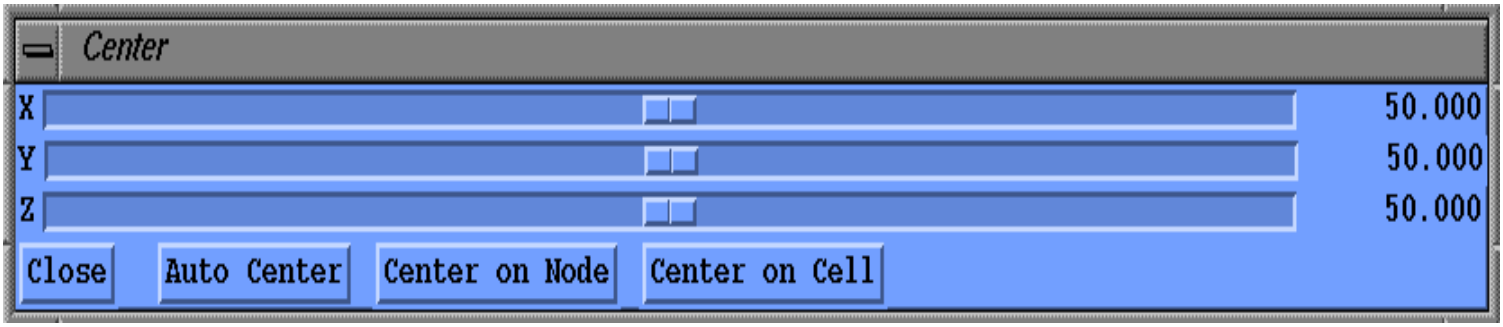


Figure 5−9. Center menu

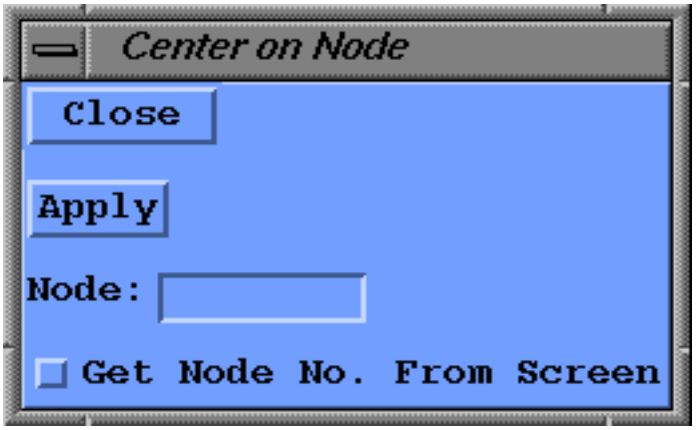


Figure 5−10. Center on node menu

slider bars labeled X, Y, and Z, respectively. Sliding these bars in either direction changes the coordinates of the center of rotation. On the right of each slider are numbers corresponding to the current coordinate for that axis. These numbers will change as the slider is moved. By default, GMV calculates the most convenient center coordinates possible. The "Auto Center" button is used to automatically center within the objects that are currently drawn. Click on "Auto Center" and see the sliders move to their new locations. To close the "Center" window, click on "Close" in the lower left corner. Auto Center is also used to center the current image.

Center on node or cell

Click on the "Center on node" button and the menu Fig. 5−10 appears. Enter a node number, or click on the "Get Node No. From Screen" toggle to set the node number with a probe. Then click on the "Apply" button to set the center on the node. The "Center on Cell" option works in the same manner as "Center on Node".

Clip:

The clip option in the Controls−1 menu controls the front and back clip planes. When selected, the Clip menu appears (see Fig.5−11). Dragging the Frontclip slider to the right causes the image to be erased from front to back by a plane parallel to the screen. The backclip erases the image from back to front.

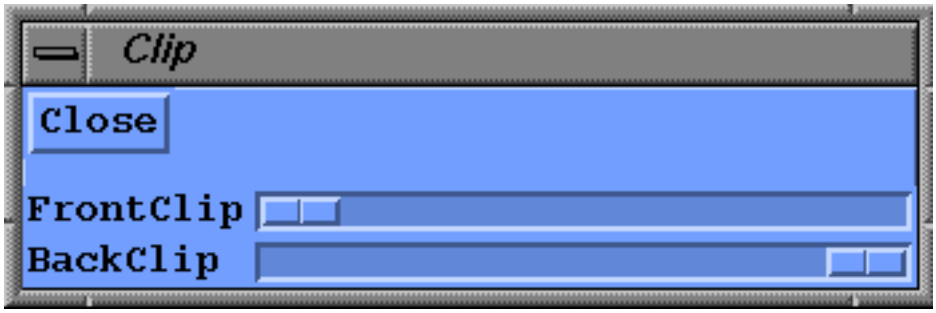


Figure 5−11. Clip slider controls

Color Bar:

The color bar option is in the Controls−1 menu. When present, the color bar, (see Fig. 5−12), is a colored scale on the far left side of the main viewer. It proceeds from blue at the low end to red on the high end and all the other colors of the spectrum in between. There are numbered tick marks down the right side of the bar. These marks tell you what each color represents numerically. The color bar will be labeled on the top as to which drawing mode (nodes, cells, cutplane, tracers) and which field it currently represents, such as (i.e. speed, pressure or temperature.)

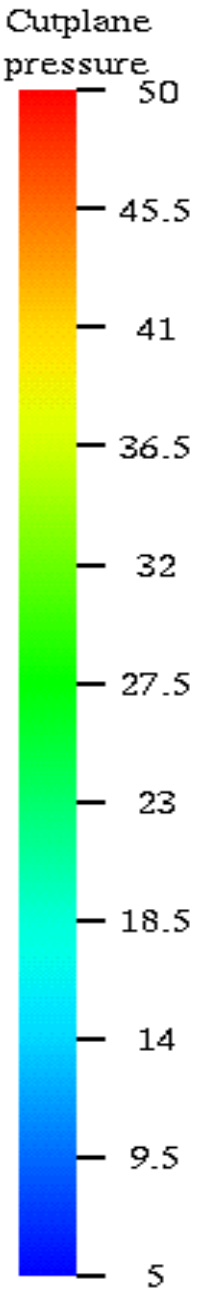


Figure 5−12. Color bar

Turning on

Click on the color bar option and a submenu will appear on the right. Two basic operations can be performed in the submenu: turning the color bar on or off, and stating a field preference. Click on the "On" box to toggle the color bar on or off.

Preferences

It is possible for several different color bar fields to be active simultaneously. For example, you may want to superimpose the tracers on a color contour plot of speed along a cutplane. GMV displays only one color bar at a time; therefore, GMV must decide which one to display. This is the purpose of the color bar preferences option. Underneath the on/off box is the word "Preferences" with a small triangle next to it. Click on the word "Preferences." A second submenu appears. This submenu has five options in it: "None," "Nodes," "Tracers," "Cells," and "Cutplane." The box next to the currently active preference will be highlighted. To select a preference, click on its corresponding box. The default preference is "None." When no preference is given, GMV uses a predetermined order of precedence. This precedence is the same as the order in the menu with the low precedence given to the first entry after "None". When a preference other than none is selected, GMV then makes the selection the highest priority.

Color Edit:

The Color Edit option is located under "Controls−1" main menu control bar. Three options are available, "Materials, Isosurfaces, Isovolume", "Field Data Colormap", and "Background

Color".

Materials, Isosurfaces, Isovolume

Select the "Materials, Isosurfaces, Isovolume" option and the menu, Fig. 5−13, appears. The square in the center of the left side of the menu contains the current color. The current color can be changed in several ways. First, change it by using the red, green, and blue color slider bars located at the bottom left of the menu. Any color can be created using a combination of these colors. The current color's transparency can also be changed with the transparency slider bar at the bottom of the menu. A transparency value of one is no transparency, and a value of zero is transparent. Secondly, there are six predefined color buttons at the upper left of the menu labeled "Copper," "Steel," "Aluminum," "Lead," "Gold," and "H. E." Clicking on any one of these changes the current color square to resemble those particular materials. For example, the "Steel" button changes the current color to a grey, steel−like color. The third way to change the current color is to click on the

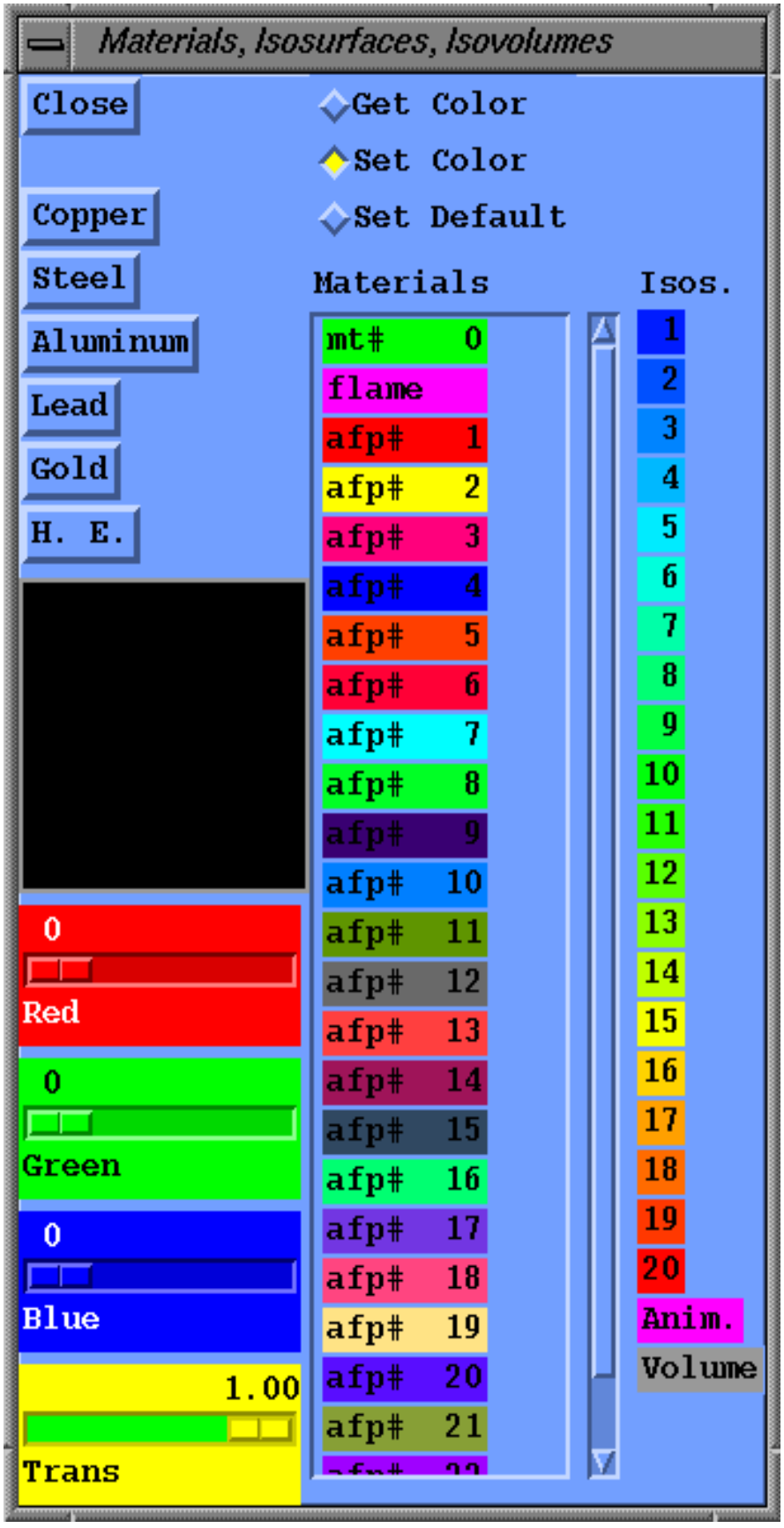


Figure 5−13. Materials, Isosurfaces, Isovolume color edit menu

Figure 5−14. Field Color Map Selection Menu

"Get Color" option in the upper right portion of the window. When "Get Color" is selected, the current color will change to the color of the material or isosurface box clicked upon. This is useful for copying colors from one place to another.

Changing material or isosurface colors

The right side of the "Color Edit" submenu is lined with material and isosurface color boxes. The color of a particular material or isosurface box is the color that the material or isosurface will appear in, in the main viewer. There is also one color box at the bottom of the list of isosurface colors labeled "Anim." This is the color used for isosurface animation. To change a material or isosurface color, first change the current color to the desired one. Next, click on "Set Color" in the upper right part of the menu. When "Set Color" is selected, any material or isosurface box you click on will change to the current color. Therefore, any material or isosurface color may be reassigned.

At the bottom right of the "Color Edit" menu is a button to change the color of an isovolume. This button works as described above; choose a color then click in the "Volume" button to set the color of the isovolume to the selected color.

Reinstating default colors

If you want to reset the GMV default colors, use the "Set Default" option. When this option is highlighted, clicking on a material or isosurface box will return that box to its default color.

Field Data Colormap

To change the colormap for field data, select the "Field Data Colormap" option and the menu, Fig. 5−14 appears, Click on one of the predefined colormaps, or click on "Read gmv colormap file" to read in a user defined colormap. The gmv

colormap file is an ASCII file with the following format. The first line contains the keyword "gmvcmap", followed by 256 lines with three floating point numbers. The three floating point numbers on each line represent the red, green, and blue components of the color. These components must be between 0. and 1. where 0. means no color and 1. is full color.

Background Color

To change the background color, select the "Background Color" option and the menu, Fig.5−15 appears. Move the sliders until the background color you wish is displayed in the color box. Then click on the "Apply" button to change the background color.

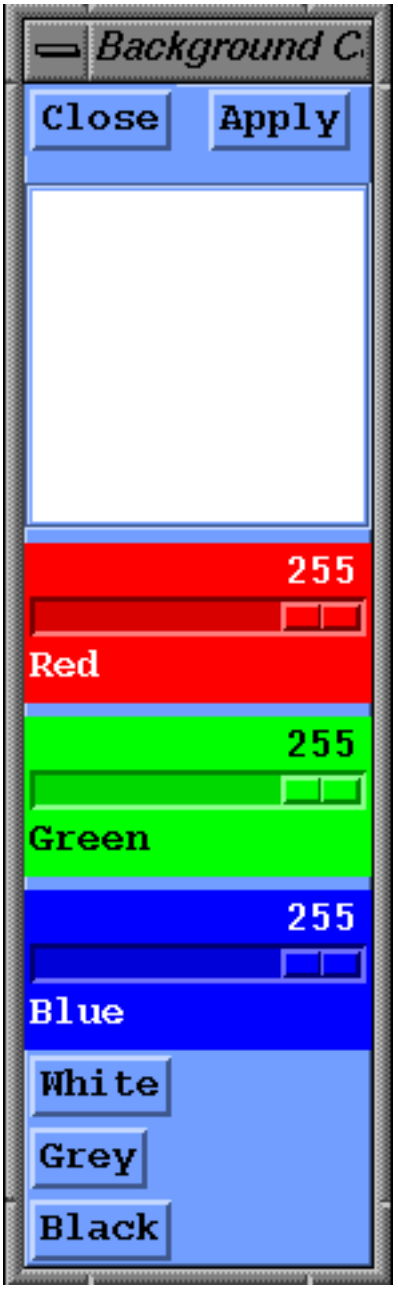


Figure 5−15. Background Color Menu

Contour Levels:

"Contour Levels" is in the "Controls−1" menu. Its function is to change the number of contour levels (lines) seen on cell faces and cutplanes. Select this option and the "Contour Levels" menu, Fig. 5−16, appears. Change the number of levels in the text box and press "Enter" to initiate the change.

Figure 5−16. Contour Levels Menu

The Ctl−2 (Controls−2) Menu

Cycle:

"Cycle" is in the "Ctl−2" menu. Its function is to toggle on or off the cycle number in the upper−left corner of the main viewer. Cycle numbers are sometimes included in input files to help you keep track of which simulation cycle generated the file. If no cycle information exists in the input file, GMV will display a zero.

Data limits:

The "Data Limits" option is in the "Ctl−2" menu. This allows you to specify the minimum and maximum values of the color bar for Node Fields, Cell Fields, and for any tracers. When the limits of the color bars are changed, the object in the main viewer will appear in colors that correspond to the new range of the color bar. Setting new field limits is extremely useful when making time sequence movies. Set the limits to the simulation minimum and maximum field values to have a consistent color range throughout the time sequence.

When you click on the "Data Limits" option button in the menu, a submenu will appear on the right. From the sub menu, choose either "Node Fields," "Cell Fields," or "Tracers." After a selection is made, additional submenus will appear, depending on your choice.

Fields

Choosing "Node Fields" or "Cell Fields" will bring up a submenu similar to the one pictured here.(see Fig. 5−14) The current field whose limits may be changed is displayed after the "Field" label. To change the current field, click on the "New Field" button to pop up the node or cell field selection menu, depending on which submenu was selected. Below the current field are four boxes in which the original and current field limits for will be displayed. The original limits are the minimum and maximum value for the field as read from the input file. By default, the current limits are set to the original ones. This ensures that the spectrum of color covers all the available data evenly. To change the current values, click on the minimum or maximum current value box and enter the new limit. For the new limits to take effect, click on "Apply" in the lower−left corner of the submenu. The "Use Nice Intervals" button changes the limits of the field and choses a nice set of intervals for the color bar and contours. The "Display as Logs" button displays the log of the data and generates a log scale. An error will occur for logs if the data minimum is zero or negative. To change the current limits back to the original ones,

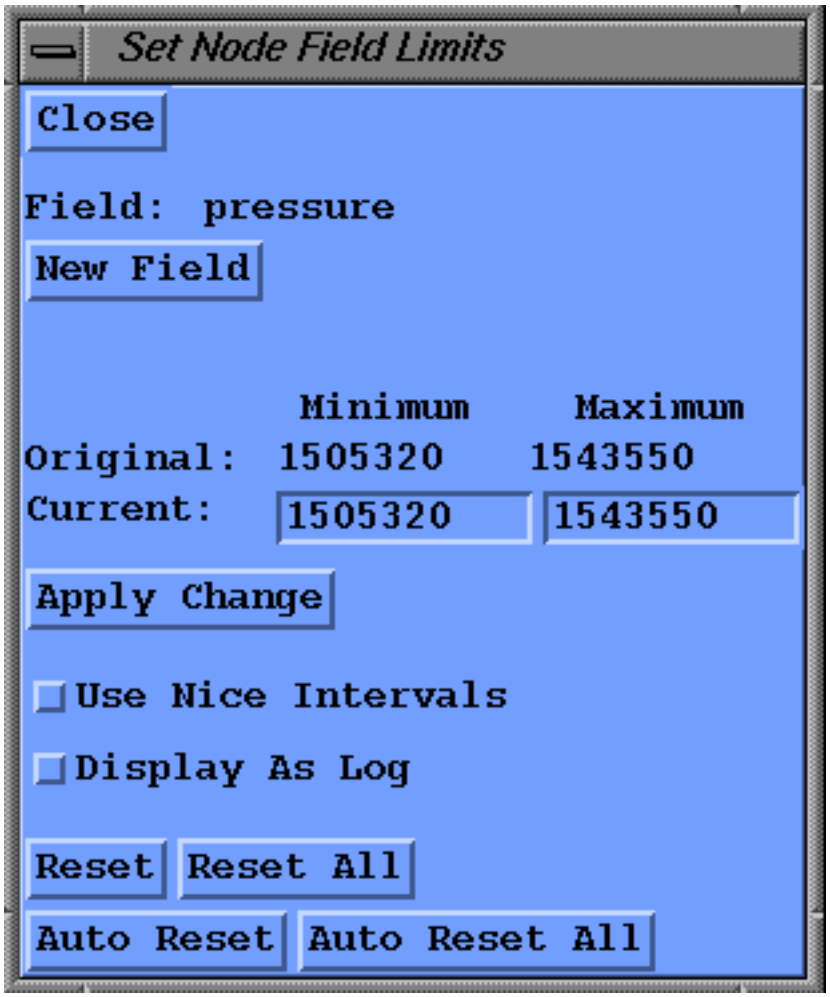


Figure 6−1. Data Limits menu

click on "Reset". To change the current limits for all fields for the selected type (nodes or cells) back to original limits, click on the "Reset All" button. The "Auto Reset" button is used with the "Auto Read" input options and with the "Put Attributes" options to automatically reset the field data limits to those on the current input file. Remember that the data limits that have been set for a field normally applies to all files read as part of an autoread or when attributes are read. The "Auto Reset All" button resets all fields for the selected type(nodes or cells) during an autoread or when attributes are read. Note that data greater then the maximum will be colored bright pink and data lower than the minimum will be colored purple. Click on "Close" in the upper−left corner of the submenu to exit the submenu.

Tracers

Choosing "Tracers" brings up a submenu labeled "Set Tracer Limits." The procedure for changing tracer limits is similar to changing node fields.

Distance Scale:

"Distance Scale" is in the "Ctl−2" menu. Its function is to toggle on or off a distance scale at the bottom of the main viewer.

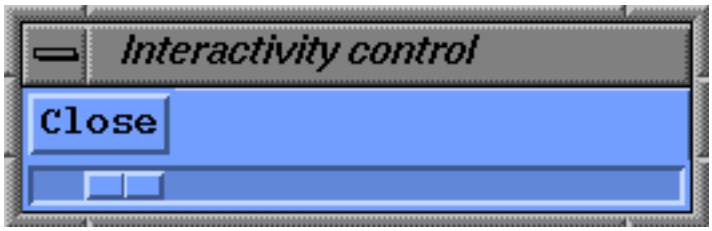


Figure 6−2. Interacitvity menu

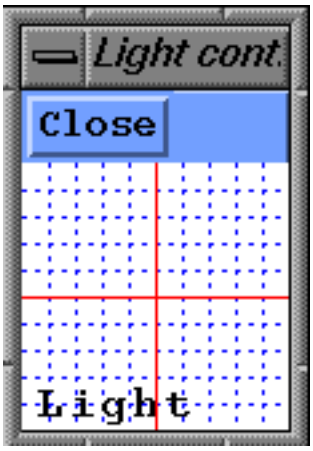


Figure 6−3. Light control menu

Interactivity:

"Interactivity" is in the "Ctl−2" menu, click on it to display the Interactivity Control menu (Fig. 6−2). Move the slider to the right to display fewer elements of the object while manipulating the image with the mouse controls, thus increasing interactivity. If you move the slider all the way to the right, a filled gridded box will appear for orientation purposes when using the mouse functions. allows user specification of the size and smoothness of lines.

Light:

The "Light" button in the "Ctl−2" menu, is used to control the light source. When selected, the Light control menus appears. By default, the light source is located at the center, change the position of the light by clicking the mouse anywhere in the light box. The crosshairs will move to the location in the light box where the current light source is located. The light source is always in front of the image and is an infinite white light with parallel rays.

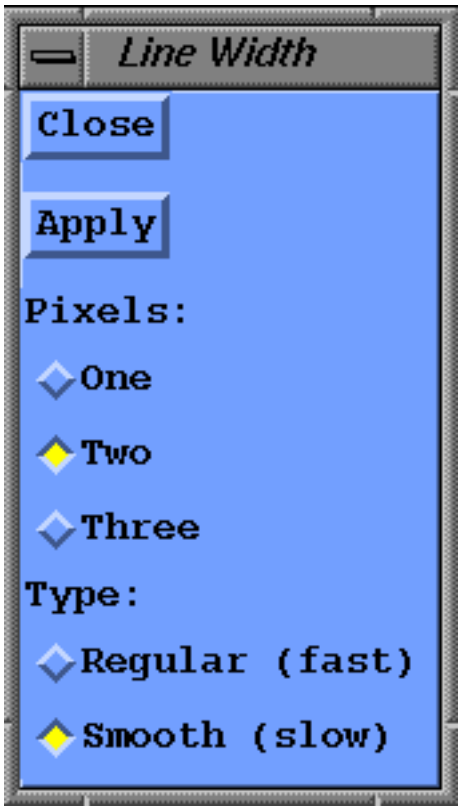


Figure 6−4. Line Width menu

Line Width:

The "Line Width" menu (Fig. 6−4) allows user specification of the size and smoothness of lines. This menu provides buttons used to select the rendering of lines using one, two or three pixels. Additionally, the particle shape may be toggled to draw smooth antialiased lines. There may not be any

difference between "Regular" and "Smooth" lines on some workstations.

Point Size:

The "Point Size" menu (Fig. 6−5) allows user specification of the size and shape of points. This menu provides buttons used to select the rendering of points using two, four, six, or eight pixels. Additionally, the point shape may be toggled to use either square or round points.

The square points update quickly; round points rely on antialiasing to generate curvature and thus update slower. Mesa OpenGL versions occasionally exhibit abnormal point updates when antialiasing is used for point generation.

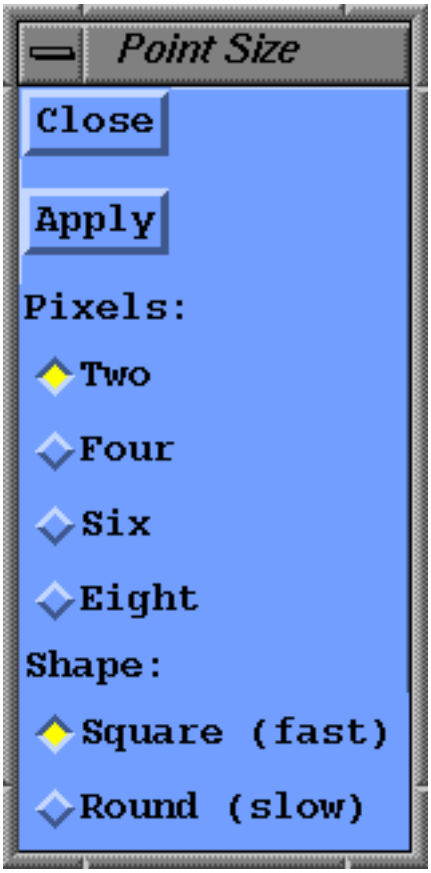


Figure 6−5. Point SIze menu

Plot box:

The plot box is the three−dimensional plotting area. By default GMV chooses dimensions for the box to enclose all the data found in the input file in the smallest possible volume. To change the dimensions of the plot box, use the "Plot Box" option found in the "Controls−2" menu.

Click on the "Plot Box" button, to open the menu. A menu labeled "Set Plot Box" will appear (see Fig. 6−6). There are boxes to define minimum and maximum values on each axis such that any length, width, or height plot box can be created. To change a value, click on the appropriate box and enter a new number. To cause the new box dimensions to take effect click on "Apply." Changing the dimensions of the plot box also affects the location of the center. The center will automatically change to correspond to the center of the plotbox,

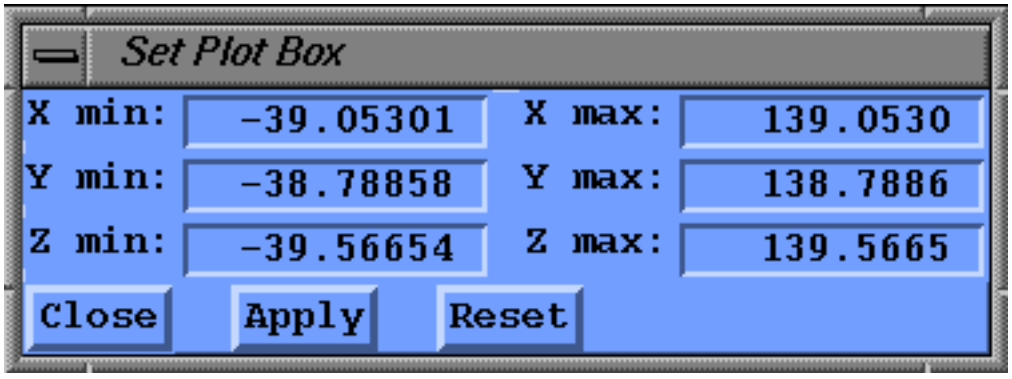


Figure 6−6. Plot Box menu

but not necessarily the center of the object, which is default. To return the center to the middle of the new plot box, open the center tool under "Controls−1" menu and click on "Auto Center" in the submenu. Click on "Close" when modifications to the plotbox are complete. Use plot box when generating time sequence animations with moving objects. Set the plot box to the simulation minimum and maximum X, Y, and Z values so that the objects move across the screen.

Scale axes:

"Scale Axes" is under the "Control−2" menu. Normally, all the axes in GMV are on an equal scale. It may become necessary to exaggerate certain features of an object by stretching it in certain directions. Changing the scale factors for the axes is done using the axes scaling tool.

To open the window, choose "Scale Axes" from the menu. A submenu similar to Fig. 6−7 appears. There are three boxes in the window, each containing a scale factor for a particular axis. The first time the submenu is opened, you will notice that all the scale factors are set to one (default setting). To change a scale factor for an axis, click on the box for that axis . Enter the new scale factor and click on "Apply" button in the lower−left corner of the submenu. GMV will redraw the object in the main viewer on the newly rescaled axes. To close the window, click on the "Close" button.

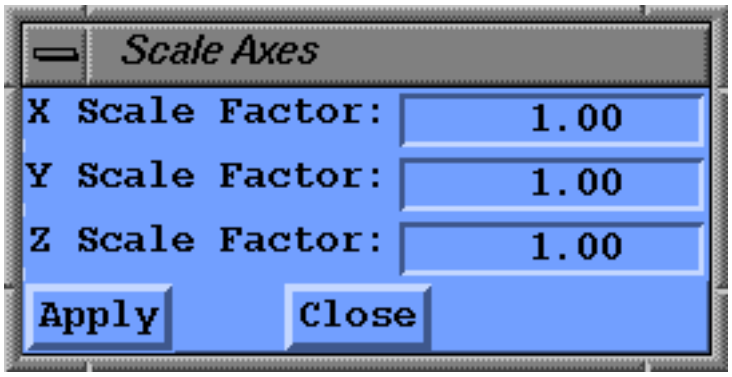


Figure 6−7. Scale Axes menu

The Ctl−3 (Controls−3) Menu

Subset:

"Subset" is in the "Ctl−3" menu. A subset is a portion of all the available data. A subset box may be defined in GMV so that nodes, cells, tracers, or polygons within that box are shown. To define a subset, choose the "Subset" option from the menu. A submenu will appear on the right. Choose either "Nodes, Cells, and Tracers" or "Polygons," depending on the type of objects being manipulated.

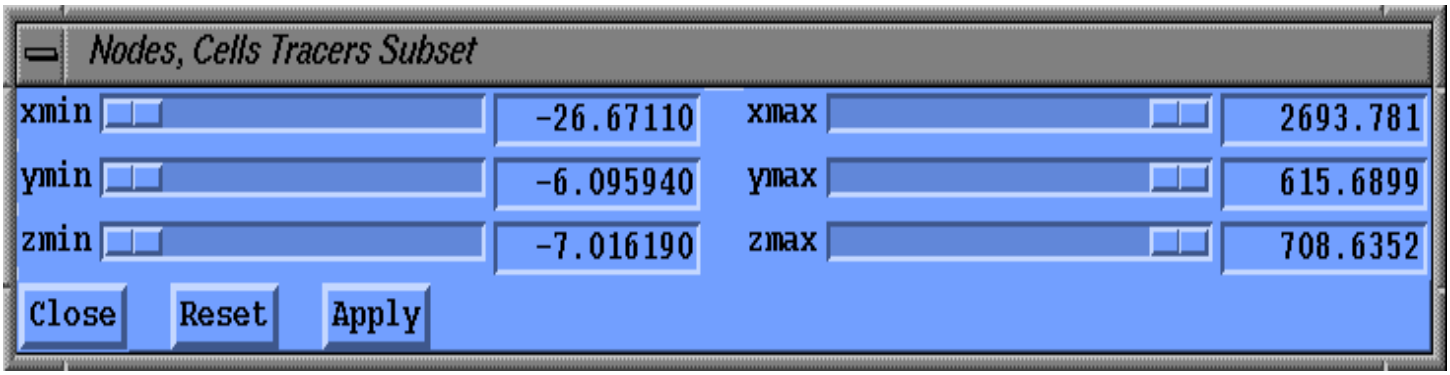


Figure 7−1. Subset menu

Nodes, cells, and tracers

When this option is selected, a submenu similar to Fig. 7−1 appears. There are six slider bars: a minimum and maximum slider for each axis. The minimum and maximum values for each axis are used to create a box that houses the subset. To change the size of the subset box, drag the sliders back and forth until satisfied with the size of the subset box. Click the "Apply" button and all nodes and tracers outside the subset box will disappear, leaving only that data inside the box. Cells whose cell centers are outside the box will not be drawn. Click on the "Reset" button to reset the subset box. To close the window, click on "Close."

Polygons

Choosing the polygons option from the "Subset" submenu brings up the same polygon subset submenu explained on page 3−10.

Texture smoothing:

The "Texture Smoothing" option in the "Ctl−3" menu enables the use of a texture map to more accurately display smooth color gradients when viewing node field data on cell faces, surface faces, isosurfaces, isovolumes, cutplanes,

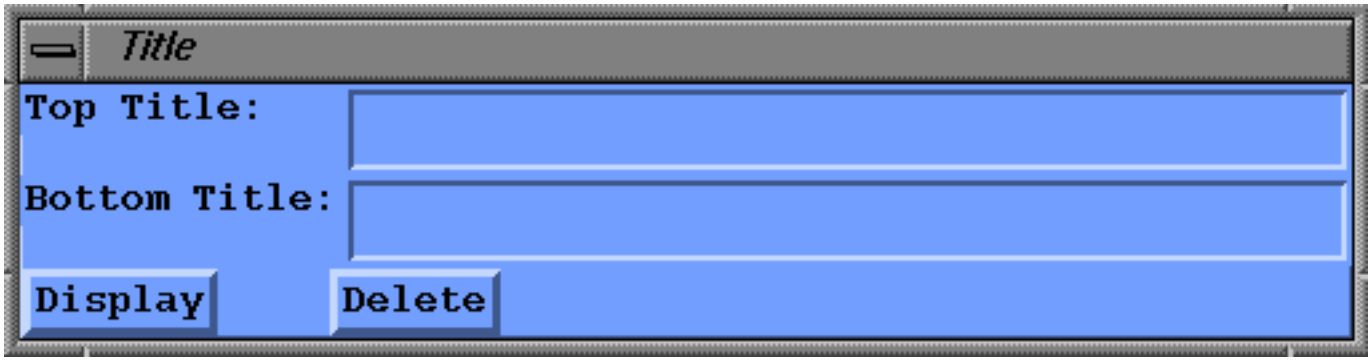


Figure 7−2. Title menu

cell edges, and surface edges. Texture smoothing may be slower on some graphics boards, but the colors are more accurate. Texture smoothing solves the problem of bad color interpolation when minimum and maximum data values exist in a single face.

Time:

A GMV data file may contain a time index. Having the time index on the screen is useful for simulations. The "Time" button in the "Ctl−3" menu is used to toggle the time display on and off. When the time is displayed, it is in the upper right corner of the main viewer. If the input file contains no time data, GMV will display a zero.

Title:

A top and/or bottom title may be added to the main main viewer. To add a title, choose "Title" from the "Ctl−3" menu. A submenu will appear (see Fig. 7−2). Enter the text of the title into the "Top Title" and/or the "Bottom Title" text boxes. Click on the "Display" button to add the title to the top center and/or bottom center portion of the main viewer. The submenu closes automatically. To remove the title, reopen the submenu and click on the "Delete" button.

Use Display List:

An option to use OpenGL Display lists is available under the "Ctl−3" menu Click on the "Use Display List" button to toggle the display list option on and off. Using the Display List option will dramatically increase the drawing speed of rotation, zoom, and pan functions when running GMV remotely. This increase in drawing speed occurs because the drawing primitives are stored in a list that exists on the display workstation; thus individual drawing commands are not delivered over the network. There may be a marginal drawing speed increase with display lists when running GMV on a local workstation, but more memory will be used. Note that when the display list option is used, the "PS Lines" Snapshot option is not available.

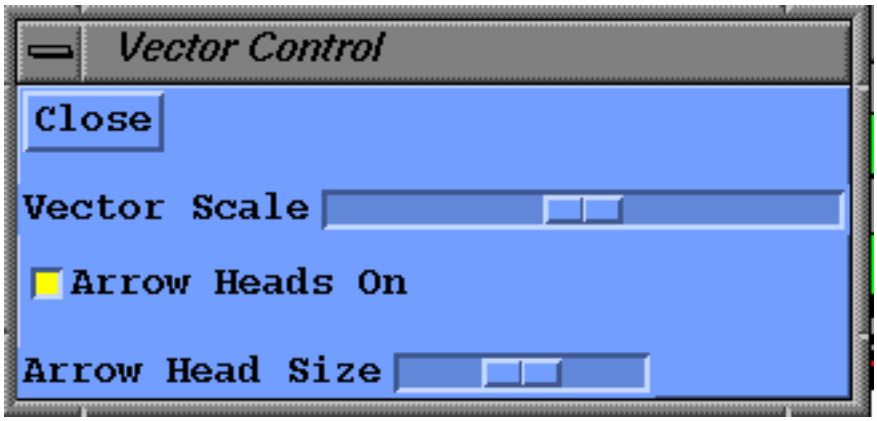


Figure 7−3. Vector Control menu

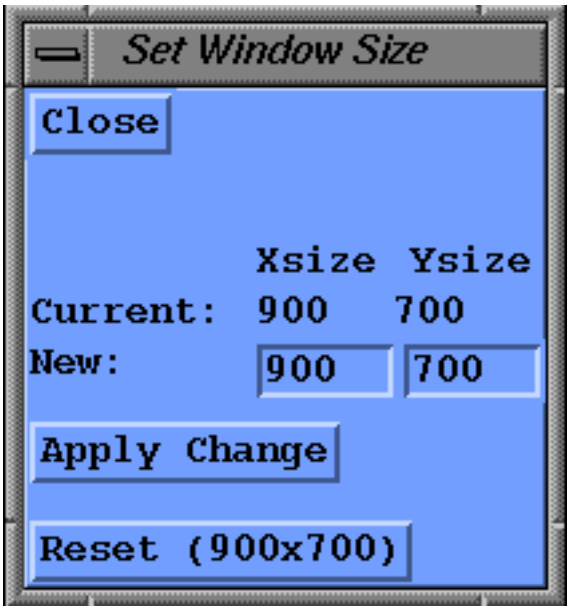


Figure 7−4. Window size menu

Vector Control:

To control the size and tip shape of vectors, select "Vector Control" from "Ctl−3" and the Vector Control menu (fig. 7−3) appears.

The "Vector Scale" slider bar controls the relative length of the current vectors. This adjustment allows you to scale the length of the vectors to your own liking. Although the length of the vectors may increase when the slider is dragged to the right, the length is always proportional to the magnitude of the data from which the vectors were drawn in the first place.

The "Arrow Heads On" toggle button determines the tip shape of the vector. If this toggle is on, an arrow head will be drawn at the tip of the vector, otherwise a text color line segment tip is drawn.

The "Arrow Head Size" slider bar controls the size of the arrow head from half the original size to one and a half times the original size.

Virtual Trackball:

The Virtual Trackball option changes the rotation motion of the left mouse button. With the virtual trackball on, the left mouse rotation control will now perform like the rotation control in most CAD systems. Pressing the left mouse button while in the interior of the display will grab and rotate the image about all three axes. Pressing the left mouse button while at the edges of the display will rotate the image about an axis normal to the screen.

Window Size:

To control the size of the drawing window, select "Window Size" from "Ctl−3" and the Window size menu (Fig. 7−4) appears. Set the window size, in pixels) on the "New Size" text boxes and press the "Apply Change" button to make

the change. Press the "Reset" button to return to the default size.

Zoom (Rubberband):

Select the "Zoom (Rubberband)" from "Ctl−3" and the Zoom Rubberband menu (fig. 7−5) appears. Use this menu to Zoom in to a user selected area in the image. To select the are to zoom into, press the "Draw Zoom Box" button. A crosshair will appear on the display window. Use the mouse to move the crosshair to one corner of the box where you want to zoom into. Press and hold the left mouse button and move the mouse until the desired zoom box is drawn. If the zoom box is not in the proper location, press the "Undo Zoom Box" button to erase the box, then press the "Draw Zoom Box" button to draw another box. Press the "Apply Zoom" button to zoom in on the box. At this time, the image will be magnified such that the largest edge of the zoom box will fill the drawing window. Also, the pan will be set to the center of the zoom box, and the rotational center will be set to the center of projection of the center of the box.

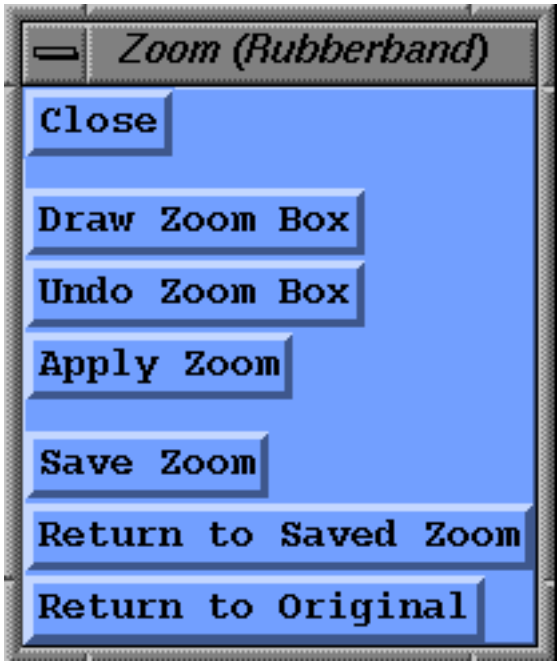


Figure 7−5 Rubberband Zoom menu

The current zoom values can be saved by clicking the "Save Zoom" button. The "Return to Saved Zoom" button will set the zoom values to those saved with "Save Zoom". Clicking on the "Return to Original" button will return the image to a magnification of 1. and to original pan and center values.

The Reflections Menu

X−axis Reflection:

Choose this button to reflect the mesh data about the X−axis.

Y−axis Reflection:

Choose this button to reflect the mesh data about the Y−axis.

Z−axis Reflection:

Choose this button to reflect the mesh data about the Z−axis.

Mirror Imaging:

A special word of caution is in order. Please note that reflecting about more than one axis is cumulative. For example, GMV, when requested to reflect about an axis, will always reflect whatever is in the main viewer, even if half that data is reflected data.

Mirror imaging is useful for problems that have a great deal of symmetry in them. The designer might want to include data for only half, a fourth, or an eighth of the problem and then create the rest of the image by reflecting the data about an axis. This method saves much time and memory when running the simulation.

The View Menu

The options in the view menu are designed to allow three different ways to project the object on the main viewer. The three different ways GMV can display objects are: orthographic, perspective, and flight. Only one can be chosen at a time.

Orthographic:

The orthographic projection is the default setting for GMV. It displays the object with parallel sides.

Perspective:

The perspective projection represents object depth on the two−dimensional screen.

The difference between orthographic and perspective becomes clearer with an example: To draw a road going off into the distance, the orthographic drawing would be two parallel lines, whereas, perspective drawing would have two lines converging on a common point where the road is no longer visible (the vanishing point).

Flight:

The "Flight" option is not really a special projection, rather a combination of the perspective view with special zooming−in capability that flies through the objects. When the flight mode is engaged, large red, green, and blue crosshairs will appear on the main viewer. These act as an aiming device. Mouse buttons operate differently in flight mode. The movements in flight mode simulate being in a helicopter. The left mouse button controls heading and pitch. Holding the left mouse button and moving left and right, changes the heading, where 0 degrees is parallel to the positive x axis. Moving up and down with the left mouse button pressed, changes the pitch angle where 90 degrees is level flight and 0 degrees is straight down. Holding the right mouse button and moving up and down, moves forward or backwards into and through objects. The middle mouse button still allows left, right, up, and down panning, with the capability of moving through objects. Thus interactive fly−throughs of GMV data from any direction are available.

Stereo Perspective:

The "Stereo Perspective" option places a local SGI workstation into full screen stereoscopic mode. You will need the "Crystal Eyes" LCD glasses and emitter (or equivalent) in order to get the stereoscopic effect. When you click on

this option, the screen will turn blank for a moment, then a full screen image will appear. Any menus that were open will still be visible, except they will be twice as long. You may have to use Alt and the right mouse button in order to move a menu. The image can be manipulated with the mouse just as in orthographic or perspective views. Three keyboard keys are used to control the stereoscopic mode. Press the x key to exit stereo mode, press the m key to increase eye separation, and press the n key to decrease eye separation. Use the eye separation controls (m and n keys) to adjust the stereoscopic image to give the best stereo effect and comfortable viewing. Note, SGI only, the stereo option can only be used on local SGI workstations; it will not set a remote monitor into stereo mode. If for some reason GMV aborts while in stereo mode, issue the following command to return your monitor back to normal mode: /usr/gfx/setmon −n 72HZ. Note, non−SGI, you must set the workstation into stereo mode through the X server before starting GMV.

Stereo Flight:

The "Stereo Flight" option places a local SGI workstation into full screen stereoscopic mode just as in "Stereo Perspective". The difference is that this option operates the same as in Flight mode.

The GMV Input Format

Input Specifications:

The format for GMV's input file follows. Please note that there are relatively few required entries, most data is optional, and keywords are used to identify its type. The data on the file can be either formatted ASCII or IEEE unformatted (but not both). Keywords are italicized and data names are in boldface. Example names for variables or flags are in double quotation marks. A description of the input line follows the data names or keywords. Only ***gmvinput***, the file type, node data, cell or face data, and ***endgmv*** are required, everything else is optional, however, except for material and velocity keywords, each keyword may be used only once. For example, a second variable list is not allowed.

For IEEE unformatted files, keywords must be written as eight character words.

gmvinput **file_type**

The first line identifies the file as a GMV input file with the file_type being either "ascii", "ieee" or "ieeei4r8".

nodes **nnodes −** Node points and number of points.

x(nnodes) − Float, x coordinates.
y(nnodes) − Float, y coordinates.
z(nnodes) − Float, z coordinates.

OR (for a structured regular brick mesh)

nodes **−1 nxv nyv nzv** − Dimensions of structured regular brick mesh.

x(nxv) − Float, x coordinates.
y(nyv) − Float, y coordinates.
z(nzv) − Float, z coordinates.

OR (for a logically rectangular brick mesh)

nodes −2 **nxv nyv nzv**
x(nxv * nyv * nzv) − Float, x coordinates.
y(nxv * nyv * nzv) − Float, y coordinates.
z(nxv * nyv * nzv) − Float, z coordinates.

nodev **nnodes −** Node points and number of points. (Instead of nodes.)

x(1) y(1) z(1) − Float, first x, y, and z coordinates.
x(2) y(2) z(2) − Float, second x, y and z coordinates.
. . .
x(nnodes) y(nnodes) z(nnodes) − Float, last x, y, and z coordinates.

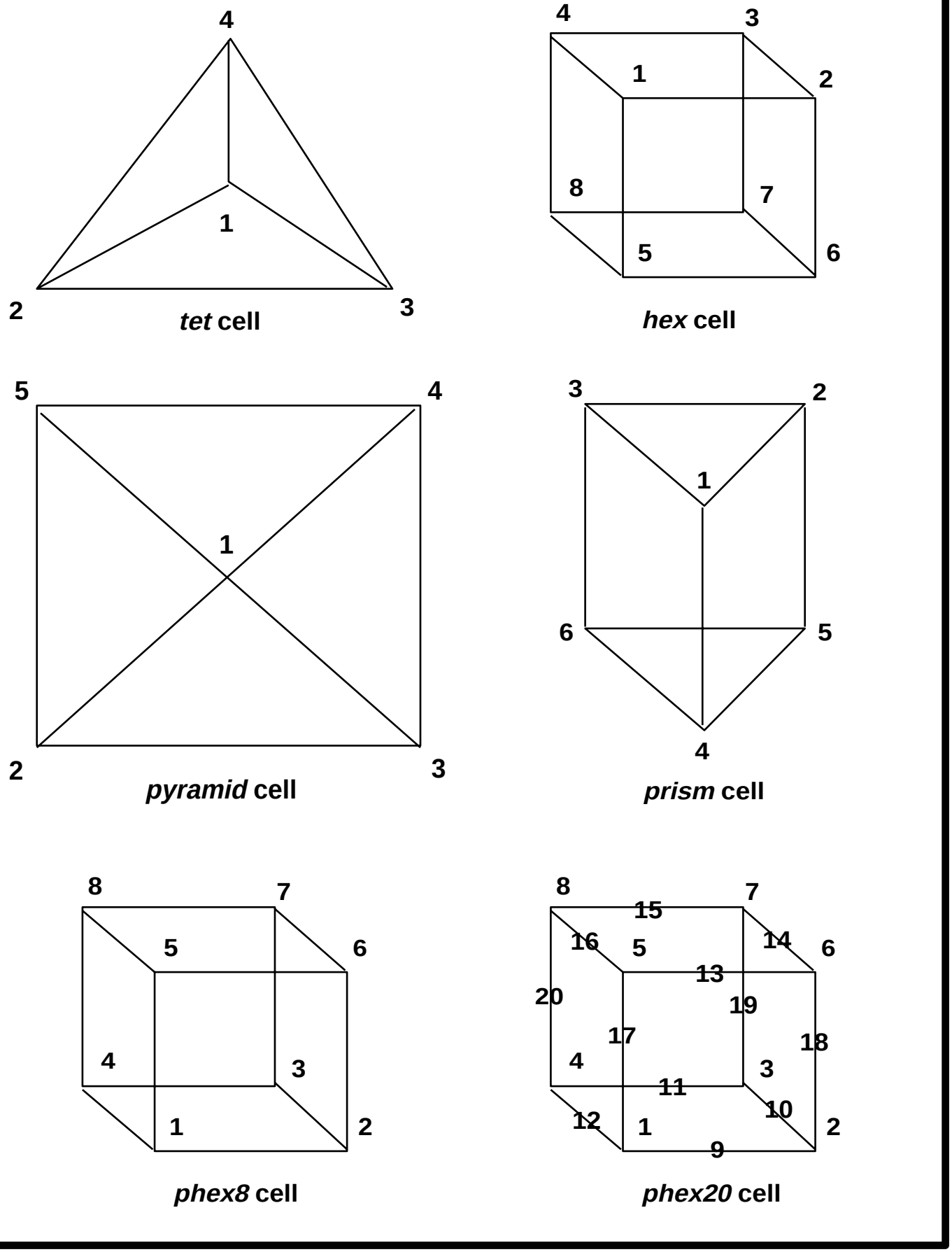


Figure 9−1. Cell vertex order

***cells* ncells −** Cell data and number of cells.

cell_type number of elements − Cell data; format depends on cell type, vertex or face data. See examples.

Allowable cell types (Note: cell types can be mixed except for vface3d and vface2d):

***general* nfaces** − General type, number of faces in cell.
nverts(nfaces) − Number of vertices per face.
vertex_ids(sum(nverts)) − Integer list of node numbers that define the polygonal faces
Note: the general type can be used to define any cell volume.

line 2 − Linear cell, with 2 vertex ids.
verts(2) − vertex ids.

***tri* 3** − Triangular cell, with 3 vertex ids.
verts(3) − vertex ids.

***quad* 4** − Quadrilateral cell, with 4 vertex ids.
verts(4) − vertex ids.

***tet* 4** − Tetrahedral cell, with 4 vertex ids.
verts(4) − vertex ids.

***hex* 8** − hexahedral cell, with 8 vertex ids.
verts(8) − vertex ids.

***phex8* 8** − hexahedral cell, with 8 vertex ids, Patran order.
verts(8) − vertex ids.

***phex*20 20** − hexahedral cell, with 20 vertex ids, Patran order.
verts(20) − vertex ids.

***prism* 6** − Prism cell, with 6 vertex ids.
verts(6) − vertex ids.

***pyramid* 5** − Pyramid cell with 5 vertex ids.
verts(5) − vertex ids.
Note: the ordering of the vertices for tet, hex, prism and pyramid is shown in Fig. 9−1. The tri and quad cells are two−dimensional entities that employ a sequential vertex numbering scheme around the periphery of the cell.

***vface3d* nfaces** − Vface3d type, number of faces in cell.
v**faces(nfaces)** − vface ids in vface keyword (see below).
Note: vface3d type cannot be mixed with other cell types.

***vface2d* nfaces** − Vface2d type, number of faces in cell.
v**faces(nfaces)** − vface ids in vface keyword (see below).
Note: vface2d type cannot be mixed with other cell types. Vface2d faces are 2D faces, i.e. edges.

vfaces **nfaces −** Vface data, number of faces.
Followed by nfaces lines of:
nverts − Number of vertices per face.
pe_no − Processor number of the face (1 if single processor).
oppface − Opposite face number.
opp_pe_no − Processor number of the opposite face (1 if single processor).
cellno − Cell number of the cell the face is part of.
vertex_ids − Integer list of node numbers that define the polygonal faces.

faces **nfaces ncells −** Face data, number of faces, and number of cells.
Followed by nfaces lines of:
nverts − Number of vertices per face.
vertex_ids − Integer list of node numbers that define the polygonal faces.
cellno1 − Cell number of the cell to the left of the face.
cellno2 − Cell number of the cell to the right of the face
The cell numbers must be between 0 and ncells where 0 indicates no cell exists on that side of the face.
Note, face data includes cell information; do NOT mix both cells and faces within one GMV input file.

nodeids − Alternate node id numbers for display
ids(nnodes) − Integer, alternate node ids.

cellids − Alternate cell id numbers for display
ids(ncells) − Integer, alternate cell ids.

faceids − Alternate face id numbers for display
ids(nfaces) − Integer, alternate face ids.

material **nmats data_type** − Material data, number of materials, and data type (0=cells, 1=nodes).
matnames(nmats) − 8 character material names.
matids(ncells) − Integer, material ids for cells.
or
matids(nnodes) − Integer, material ids for nodes.

velocity **data_type** − Velocity data, and data type (0=cells, 1=nodes, 2=faces)
u(ncells) |
v(ncells) | For cells.
w(ncells) |
or

u(nnodes) |
v(nnodes) | For nodes.
w(nnodes) |
or
u(nfaces) |
v(nfaces) | For faces.
w(nfaces) |

variable − Keyword indicating that other cell or node data sets follow. The data sets have the form:

"anyname" data_type − An eight character name for the data, the data type (0=cells, 1=nodes, 2=faces)

data(ncells, nnodes or nfaces) − array of float data.

Examples (DO NOT use quotes in actual file):
"density" 0
density_data(ncells)

"temp" 0
temp_data(ncells)

"pressure" 1
pressure_data(nnodes)

"flow" 2
flow_data(nfaces)

endvars − Keyword indicating end of variable data.

subvars − Keyword indicating that other cell or node data sets for a subset of the mesh follows. The data sets have the form:

"anyname" data_type **num_elem** − An eight character name for the data, the data type (0=cells, 1=nodes, 2=faces), the number of elements in the subset.

elem_ids(num_elem) − array of integer cells, nodes or faces in the subset.

data(num_elem) − array of float data.

Examples (DO NOT use quotes in actual file):
"bdryt" 0 10
5 6 9 13 28 101 150 181 210 300
0.5 0.5 1.2 3.4 6.1 0.5 1.8 0.1 −2.2 −3.3

"intp" 1 5
1 2 3 6 7
9.8 10.0 11.0 0.2 2.2

"facefl" 2 8
20 30 40 50 60 70 90 100
100.0 200.0 300.0 −10.0 12.2 8.5 −2.8 0.0

endsubv − Keyword indicating end of subvars data.

flags − Keyword indicating that selection flag data sets follow.

The data sets have the form:

"anyname" ntypes data_type − Flag name, number of flag types, and data type (0=cells, 1=nodes).

flagnames(ntypes) − 8 character flag type names.

iflag(ncells) − Integer, flag ids for cells.

or

iflag(nnodes) − Integer, flag ids for nodes.

Examples:
"nodetype" 4 1
"inactive" "interior" "interfac" "boundary"
node_data(nnodes)

"cnstrain" 3 0
"static" "piston" "air"
cnst_data(ncells)

endflag − Keyword indicating end of flag data.

polygons − Keyword indicating surface polygon data follows. (eg. interface or boundary faces.)

material_no nverts x(nverts) y(nverts) z(nverts)

Where:

material_no − Integer number related to material data.

nverts − No. of vertices.

x(nverts) − x coordinate of polygon vertices.

y(nverts) − y coordinate of polygon vertices.

z(nverts) − z coordinate of polygon vertices.

This data is repeated for all polygons.

endpoly − Keyword to indicate end of polygon data.

tracers **ntracers** − Tracer points and the number of tracers input.
x(ntracers) − Float, x coordinates.
y(ntracers) − Float, y coordinates.
z(ntracers) − Float, z coordinates.
Followed by trace data of the form:
"anyname" − An eight character name for the data.
data(ntracers) − array of float data.

Examples:
"temp"
temp_data(ntracers)

"pressure"
pressure_data(ntracers)

endtrace − Keyword indicating end of variable data.
traceids − Alternate tracer id numbers for display
ids(ntracers) − Integer, alternate tracer ids.

probtime **ptime** − Keyword and floating point problem time value.

cycleno **cycleno** − Keyword and integer problem cycle number.

surface **nsurfaces −** Surface facet data and number of facets.
Followed by nsurfaces lines of:
nverts − Number of vertices per facet.
vertex_ids − Integer list of node numbers that define the polygonal facet.

surfmats − Surface material data.
matids(nsurfaces) − Integer, material ids for surface facets.
Note, surface data must be input before surfmats.

surfvel − Surface velocity data.
u(nsurfaces)
v(nsurfaces)
w(nsurfaces)

Note, surface data must be input before surfvel.

surfvars − Keyword indicating that other surface field data sets follow. The data sets have the form:

"anyname" − An eight character name for the data.

data(nsurfaces) − array of float data.

Examples (DO NOT use quotes in actual file):
"density"
density_data(nsurfaces)

"temp"
temp_data(nsurfaces)

endsvars − Keyword indicating end of variable data

surfflag − Keyword indicating that surface selection flag data sets follow. The data sets have the form:

"anyname" ntypes − Flag name and number of flag types,.

flagnames(ntypes) − 8 character flag type names.

iflag(nsurfaces) − Integer, flag ids for surfaces.

Note, surface data must be input before surfmats.

Examples:
"surftype" 4
"inactive" "interior" "interfac" "boundary"
sflg_data(nsurfaces)

"cnstrain" 3
"static" "piston" "air"
cnst_data(nsurfaces)

endsflag − Keyword indicating end of flag data.

surfids − Alternate surface id numbers for display

ids(nsurfaces) − Integer, alternate surface ids.

groups − Keyword indicating that user defined group sets follow. The data sets have the form:

"anyname" − An eight character name for the group,

data_type − The Data type (0=cells, 1=nodes, 2=faces, 3=surfaces),

nelem −The number of elements in the group.

data(nelem) − array of integer cells, nodes or faces, depending on type.

Examples (DO NOT use quotes in actual file):
"cells1" 0 5 1 4 7 10 12
"nodes1" 1 8 5 7 10 30 45 50 55 62
"faces1" 2 4 1 4 12 18

endgrp − Keyword indicating end of group data

comments − Keyword indicating that ASCII comments follow.

endcomm − Keyword indicating end of comments.

codename "**anyname**" − The name of the code that generated the file where "anyname" is the eight character name of the code.

codever "**version**" − The version of the code that generated the file where "version" is the eight character version of the code.

simdate "**date**" − The date the file was generated where "date" is in the form mm/dd/yy.

endgmv − Keyword signifying the end of the input file.

Input Data Details:

Header

The header line contains the "gmvinput" keyword and the character variable file_type which contains either "ascii", "ieee" or "ieeei4r8. The ASCII file type indicates that the file was written as a formatted ASCII file; the file will be read using list−directed I/O, so there must be at least one space between data elements. The IEEE file type indicates the file was written as an unformatted file with IEEE single−precision floating point, character, and 32−bit integers. IEEEI4R8 indicates that all floating point data is double−precision. When writing IEEE binary files, be sure that all keywords and character data must be eight characters long.

Nodes

The "nodes" keyword describes the beginning of cell node data points and the variable nnodes on this line are the number of nodes (i.e. the length of the node data arrays that follow). The next three lines are the three floating point arrays that represent the X, Y, and Z coordinates of the nodes.

The ***nodes*** keyword has three alternate forms. The first is used to generate a structured, regular brick mesh. Entering **−1** for the number of nodes signifies this alternate syntax. After **−1** on the same line are the dimensions of the mesh; first the number of nodes along the X−axis, then the number along Y, and the number along the Z−axis. The three lines that follow contain the X, Y, and Z coordinates of the nodes along each axis, which will be used by GMV to generate the entire mesh. Note: because GMV uses this information to generate a large mesh of cells, the number of cells specified with the ***cells*** keyword must be zero.

The second alternate syntax for the ***nodes*** keyword is used to generate a logically rectangular structured mesh. Entering −2 for the number on nodes signifies this alternate syntax. After −2 and on the same line are the dimensions of the mesh; first the number of nodes along the X−axis, then the number along Y, and the number along the Z−axis. The three lines that follow contain the X,Y, and Z coordinates of the nodes for all nodes (nx*ny*nz), which will be used by GMV to generate the entire mesh. Note: because GMV uses this information to generate a large mesh of cells, the number of cells specified with the ***cells*** keyword must be zero. For any meshes that are closed, you need to repeat the necessary nodes for closure.

Node data input may also be performed by consulting a remote file (see the ***fromfile*** description below).

Nodev

The "nodev" keyword is an alternate form of the "nodes" keyword. Use nodev to input the node coordinates as a triplet on each line. There will be nnodes lines of the x, y and z coordinate on each line. The nodev keyword cannot process a structured brick mesh.

Cells

The ***cells*** keyword indicates the beginning of cell descriptions. The variable ncells on this line are the number of cell descriptors that follow. There are six standard cell types that GMV can read, ***line, tri, quad, tet, hex, pyramid, prism, phex8,*** and ***phex20***. The type is followed by the number of vertices contained in the cell, 2 for line, 3 for tris, 4 for quads, 4 for tets, 8 for hex, 5 for pyramid, 6 for prisms, 8 for phex8, and 20 for phex20. The next line contains the node numbers for the cell vertices. The vertex ordering for selected standard cell types is shown in Fig. 9−1. This ordering must be followed in order to ensure outward pointing normals. The tri and quad cells are simple two−dimensional entities that employ a sequential vertex numbering scheme counterclockwise around the periphery of the cell.

The ***general*** cell type is available for nonstandard cells. These cells are described by their faces. The nfaces variable indicates the number of faces for the cell. The next line of data is the number of vertices for each cell face. The third line

of the set contains the node numbers of the vertices for each face for all faces. The integer array size for the nodes will be the sum of the vertices for the cell faces. The faces do not have to be specified in any order. However, the vertices for each face must be specified in an order that describes the face polygon and generates an outward normal using the right hand rule..

The ***vface3d*** cell type is available for nonstandard 3D cells. These cells are described by their face numbers within the vface keyword. The nfaces variable indicates the number of faces for the cell. If ***vface3d*** is used, all cell types must be ***vface3d***.

The ***vface2d*** cell type is available for nonstandard 2D cells. These cells are described by their face numbers within the vface keyword. The nfaces variable indicates the number of faces for the cell. If ***vface2d*** is used, all cell types must be ***vface2d***. The faces in ***vface2d*** are 2d faces (edges).

The node and cell data are required and must be in order, although the number of cells can be zero if no cells exist. Note, there is no external numbering for the nodes and cells; the order of input is the numbering sequence for both nodes and cells.

Cell data input may also be performed by consulting a remote file (see the ***fromfile*** description below).

Vfaces

The ***vfaces*** keyword indicates the beginning of polygonal face descriptions related to the ***vface3d*** or ***vface2d*** cell types. The variable ***nfaces*** on this line is the number of face descriptors that follow. There are nfaces descriptor lines that contain the number of vertices in the face, the processor number for the face, the opposite face number, the processor number for the opposite face, the cell in which this face is part of, and the list of node numbers of the vertices for the face. Order the face vertices to generate an outward normal using the right hand rule. If the face describes a ***vface2d*** face, then it must contain only two vertices.

Vface data input may also be performed by consulting a remote file (see the ***fromfile*** description below).

Faces

The ***faces*** keyword indicates the beginning of polygonal face descriptions and their associated cells. The variable ***nfaces*** on this line is the number of face descriptors that follow, and the variable ***ncells*** is the number of cells in the problem There are nfaces descriptor lines that contain the number of vertices in the face, the nodes that define the face, the cell that is to the left of the face, and the cell to the right of the face. Each polygonal face can be a part of up to two cells, if the face is part of only one cell, a cell number of 0 (zero) must be input.

Face data input may also be performed by consulting a remote file (see the ***fromfile*** description below).

The ***cells*** keyword and the ***faces*** keyword must NOT both exist in the same GMV input file since they are alternate forms of the same mesh information.

Nodeids

The keyword ***nodeids*** indicates that an optional list of alternate node id numbers follows. These alternate id numbers are used for display and reference purposes within GMV. Enter nnode integers.

Cellids

The keyword ***cellids*** indicates that an optional list of alternate cell id numbers follows. These alternate id numbers are used for display and reference purposes within GMV. Enter ncell integers.

Faceids

The keyword ***faceids*** indicates that an optional list of alternate face id numbers follows. These alternate id numbers are used for display and reference purposes within GMV. Enter nface integers.

Materials

The keyword ***material***, denoting material data, is an optional but highly recommended input data type. Up to 128 materials are allowed. On the keyword line are the variables nmats (1 to 128) and data_type (0 means the material data is cell centered and 1 means the material data is node centered). The next line of data is the eight character names given to the nmats materials. Finally, the last line of material data is the cell or node centered material ids; this is an integer array. Material data is necessary if surface polygons exist.

Up to two material keywords can be entered, one for cells and one for nodes. However, both keywords must contain the same material names in the same order.

Material data can be used to distinguish between different classes of cell or node data besides the normal engineering material definitions. For example, the material data can be density layers for an ocean model, horizons in seismic data, or rock layers in a reservoir model.

Material data input may also be performed by consulting a remote file (see the ***fromfile*** description below), but only if the remote file contains only one ***material*** keyword.

Velocities

The keyword ***velocity*** indicates that optional velocity data follows. Again the **data_type** value of 0 indicates cell−centered velocities, a value of 1 indicates node−centered velocities and a value of 2 indicates face−centered velocities. The next three lines of data are the u (x component), v (y component),

and w (z component) velocity floating point arrays. Cell−centered velocities will be averaged and saved as node−centered velocities. Also, speed and kinetic energy variable fields will be automatically generated and added to the end of the input variable fields. Face centered velocities can only be entered when faces or vfaces are used to define cells.

Up to three velocity keywords can be entered, one for cells, one for nodes, and one for faces.

Variable data fields

The ***variable*** keyword is used to denote the beginning of any other cell, node or face data fields. The data are entered as a group for each field variable. Up to 250 different field variables are allowed, and each field variable is named by the user. The ***endvars*** keyword is used to end the field data input. Each field data variable is defined by two input lines. The first line contains the eight character name of the variable and the data_type of the field (0−cell data, 1−node data, 2 − face data). The second line is the floating point array for the cell or node data. Cell−centered field data will be averaged and stored as node−centered data. Face−centered data can only be entered when faces or vfaces are used to define cells.

Subset variable data fields

The ***subvars*** keyword is used to denote the beginning of cell, node or face data fields that are defined for a subset of the mesh. The data are entered as a group for each field variable for the defined elements. These fields are added to the variable field list. The ***endsubv*** keyword is used to end the field data input. Each field data variable is defined by three input lines. The first line contains the eight character name of the variable, the data_type of the field (0−cell data, 1−node data, 2 − face data) and the number of elements to define and set data for. The second line is the list of elements (nodes, cells or faces) that will carry the field data. The third line is the floating point array for the field data. Elements not defined in the list will carry a value that is less than the minimum value entered. Face centered data can only be entered when faces or vfaces are used to define cells.

Selection Flags

The ***flags*** keyword means that integer selection flag data sets follow. Up to 10 different types of selection flags and up to 128 different flag values per flag are allowed. These data sets can be any type of integer data that can be used to select a node or cell for display purposes. The names for the flags and for the flag types are placed in selection buttons in a menu. The integer data must be a number between 1 and ntypes (1 to 128). The ***endflag*** string ends the flag data.

Flag data input may also be performed by consulting a remote file (see the ***fromfile*** description below).

Polygons

The ***polygons*** keyword indicates that surface polygons data follows. The surface polygons can be interface or boundary polygons for a material. Each line describes one polygon. The line contains the material number (1 to **nmats**) associated with the polygon, the number of vertices in the polygon and the x, y, z arrays that define the vertices for the polygon. The ***endpoly*** string terminates the polygon data.

The polygons keyword can be used to describe any surface a simulation can generate. Be sure to give each surface a material number and that this material number has a material name listed under the ***materials*** keyword.

Polygon data input may also be performed by consulting a remote file (see the ***fromfile*** description below). .

Tracers

The ***tracers*** keyword indicates that tracer particle data (or any point data other than node data) follow. The ntracers variable following the keyword is the number of tracers that are input. The next three lines are the x, y, and z floating point coordinates of the tracers. Following the coordinates are the variable data fields for the tracers. The data is entered as a group for each field variable. Up to 40 different field variables are allowed. Each tracer field data variable contains an eight character variable name followed by a floating point data array. The ***endtrace*** string terminates the tracer data.

Traceids

The keyword ***traceids*** indicates that an optional list of alternate tracer id numbers follows. These alternate id numbers are used for display and reference purposes within GMV. Enter ntracer integers.

Problem Time

The ***probtime*** keyword is followed by a floating point number that represents the simulation problem time. This value is displayed at the top right corner of the main viewer.

Cycle Number

The ***cycleno*** keyword is followed by an integer number that represents the familiar cycle number. This value is displayed at the top left corner of the main viewer.

Surface

The ***surface*** keyword indicates the beginning of polygonal facet descriptions of surfaces. The variable ***nsurface*** on this line is the number of facet

descriptors that follow. There are nsurface descriptor lines that contain the number of vertices in the facet and the nodes that define the facet,

Surface materials

The keyword ***surfmat***s, denoting surface material data, is an optional but highly recommended input data type. Up to 128 materials are allowed. After the keyword line of surface material data is the surface facet centered material ids; this is an integer array.

Surface velocities

The keyword ***surfvel*** indicates that optional surface velocity data follows. The next three lines of data are the u (x component), v (y component), and w (z component) surface velocity floating point arrays. Speed and kinetic energy variable fields will be automatically generated and added to the end of the surface input variable fields.

Surface variable data fields

The ***surfvars*** keyword is used to denote the beginning of any other surface data fields. The data are entered as a group for each field variable. Up to 100 different field variables are allowed, and each field variable is named by the user. The ***endsvars*** keyword is used to end the field data input. Each field data variable is defined by two input lines. The first line contains the eight character name of the variable. The second line is the floating point array for the surface data.

Surface Selection Flags

The ***surfflag*** keyword means that integer selection flag data sets follow. Up to 10 different types of selection flags and up to 128 different flag values per flag are allowed. These data sets can be any type of integer data that can be used to select a node or cell for display purposes. The names for the flags and for the flag types are placed in selection buttons in a menu. The integer data must be a number between 1 and ntypes (1 to 128). The ***endsflag*** string ends the surface flag data.

Surfids

The keyword ***surfids*** indicates that an optional list of alternate surface id numbers follows. These alternate id numbers are used for display and reference purposes within GMV. Enter nsurface integers.

Element groups

The ***groups*** keyword is used to denote the beginning sets of arbitrary elements. A group is a named set of nodes, cells, faces, or surfaces. Up to 1000

different groups per type are allowed, and each group is named by the user. The ***endgrp*** keyword is used to end group input. Each group is defined by an eight character name, the element type (node, cell, face, or surface), the number of elements in the group and the list of element numbers.

Comments

The ***comments*** keyword means that ascii comments follow. The ***endcomm*** string ends the comments. Be sure that there is a blank before and after endcomm.

Codename, codever, simdate

The ***codename, codever, and simdate*** keywords allows the name and version of the code, as well as the date the simulation generated the file to be added to the file for identification purposes. These keywords can be placed before the ***nodes*** keyword.

Reading some GMV data from a different file:

Certain GMV keywords and their data can be read in from a different GMV input file. These keywords are ***nodes***, ***cells***, ***faces***, ***material***, ***flags***, ***polygons, nodeids, cellids*** and ***faceids***. The format for all the keywords is similar to the following:

nodes fromfile "**filename"**

This syntax is used within the scope of the current GMV file to instruct GMV that keyword data is located in a *fromfile* specified by "**filename"**. **Filename** is a user supplied character string that must be enclosed by double quotes. The use of this keyword form specifies that the fromfile will contain the pertinent data in the same format and context as would be used in the main GMV file; the fromfile must be a valid GMV format file. When GMV encounters this keyword form, main file processing stops, and the fromfile is opened and searched for the applicable data. Once the data is input, the fromfile is closed, and main GMV file processing continues.

Fromfiles are useful within GMV for displaying animation sequences and the production of movies. In many animation sequences, much of the data remains unchanged between frames (for example, nodes, cells, faces, material, flags, and polygon data may remain constant if the problem domain and physical geometry of the problem does not change between frames). Constructing distinct and complete GMV files of each frame consumes much disk space needlessly; the fromfile capability allows the placement of constant data (i.e., one or more of node, cell, face, material, flag, and polygon data) within a single file that will be repeatedly be referenced by several GMV input files. This constant file is tagged a fromfile in

this implementation.

Sample input data:

The following is a sample GMV input file in ASCII format. It includes most of the features and commands GMV allows. When read in, the file creates a cube with several other different cell types attached to it. The additional cells are: one tetrahedral cell, one prism cell, one pyramid cell, and one general cell. The general cell has ten faces, and could be described as an octagonal prism. There is a large cube constructed from square polygons, each with a different material, that encloses all of the cells. In addition, variables, tracers, and flags with arbitrary data have been included so that you may see the format for entering such elements into an input file. The keywords are italicized. In addition, blank lines have been inserted between major elements of the file for clarity, but these are not necessary in a real input file. The input data follows:

gmvinput ascii

comments
A set of comments
endcomm

nodes 28
0 50 50 0 0 0 50 50 25 25 25 25 50 50 50 50 50 50 50 50 80 80
80 80 80 80 80 80
0 0 0 0 50 50 50 50 75 0 50 −25 16.7 33.4 50 50 33.4 16.7 0 0 16.7 33.4
50 50 33.4 16.7 0 0
50 50 0 0 50 0 0 50 25 80 80 65 0 0 16.7 33.4 50 50 33.4 16.7 0 0
16.7 33.4 50 50 33.4 16.7

cells 5

hex 8
1 2 3 4 5 8 7 6

pyramid 5
9 5 6 7 8

prism 6
10 1 2 11 5 8
tet 4
12 1 2 10

general 10
8 8 4 4 4 4 4 4 4
13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28 19 27 28 20 20 28 21 13 13 21
22 14 14 22 23 15 23 24 16 15 24 25 17 16 25 17 18 26 26 18 19 27

material 6 0
mat1
mat2
mat3
mat4
mat5
mat6
1 2 3 4 5

polygons
1 4 −100 100 100 −100
100 100 −100 −100
100 100 100 100
2 4 −100 −100 100 100
−100 100 100 −100
−100 −100 −100 −100
3 4 −100 100 100 −100
100 100 100 100
100 100 −100 −100
4 4 −100 100 100 −100
−100 −100 −100 −100
100 100 −100 −100
5 4 100 100 100 100
−100 100 100 −100
100 100 −100 −100
6 4 −100 −100 −100 −100
−100 −100 100 100
100 −100 −100 100
endpoly

tracers 10
0 20 40 60 80 100 120 140 160 180
0 20 40 60 80 100 120 140 160 180
0 20 40 60 80 100 120 140 160 180
pressure
0 5 10 15 20 25 30 35 40 45

temp
45 40 35 30 25 20 15 10 5 0
density
0 5 10 15 20 25 30 35 40 45

endtrace

traceids
2 4 10 12 15 18 20 21 22 23

velocity 0
0 0 5 0 0
0 5 5 0 10
5 5 5 10 10

variable

pressure 0
0 5 10 15 20

temp 0
0 5 10 15 20

density 0
0 5 10 15 20

endvars

flags

flagtype 3 0
good bad ugly
1 2 3 2 1

stufftype 3 0
bing bang boom
1 2 3 2 1

endflag

endgmv

Making Movies With GMV

GMV command line options:

GMV has several command line options which can be used to create time sequence movies of mesh data, noninteractively. In this manner, GMV can be included in shell scripts that create snapshots. Some of the options can be input when starting GMV interactively. The following are the available GMV command line options:

window size: ***−w xloc yloc width height***

xloc, yloc: X and Y coordinates of lower left corner of window where GMV will draw the image.
width, height: the width and height in pixels where GMV will draw the image.

input file name: ***−i filename***

Specifies an input file for GMV to read.

attributes file name: ***−a attribute_filename***

Specifies an attributes file for GMV to use to draw the image.

movie mode (noninteractive): ***−m***

Starts GMV noninteractively solely for the purpose of creating snapshots; therefore no main GMV window will appear. The only indication that GMV is actually doing something will be GMV's usual status messages printed in the shell.

snapshot: ***−s snapshot_filename***

Takes a snapshot of the object in the input file saves it in the speicfied file as SGI RGB format and is used only with the **−m** option. "AzsnapgmvAz" is the default filename.

Here is an example script that uses GMV's command line options to create a series of snapshots for use in movie making:

```
#
set verbose
set k=(1−21)
foreach i (k)
/usr/local/bin/gmv −m −a iso.attr −w 639 0 640 512 −i gmvout3.$i −s ksbmiso.$i.rgb
end
```

Of course, you could just use the movie utility described below instead.

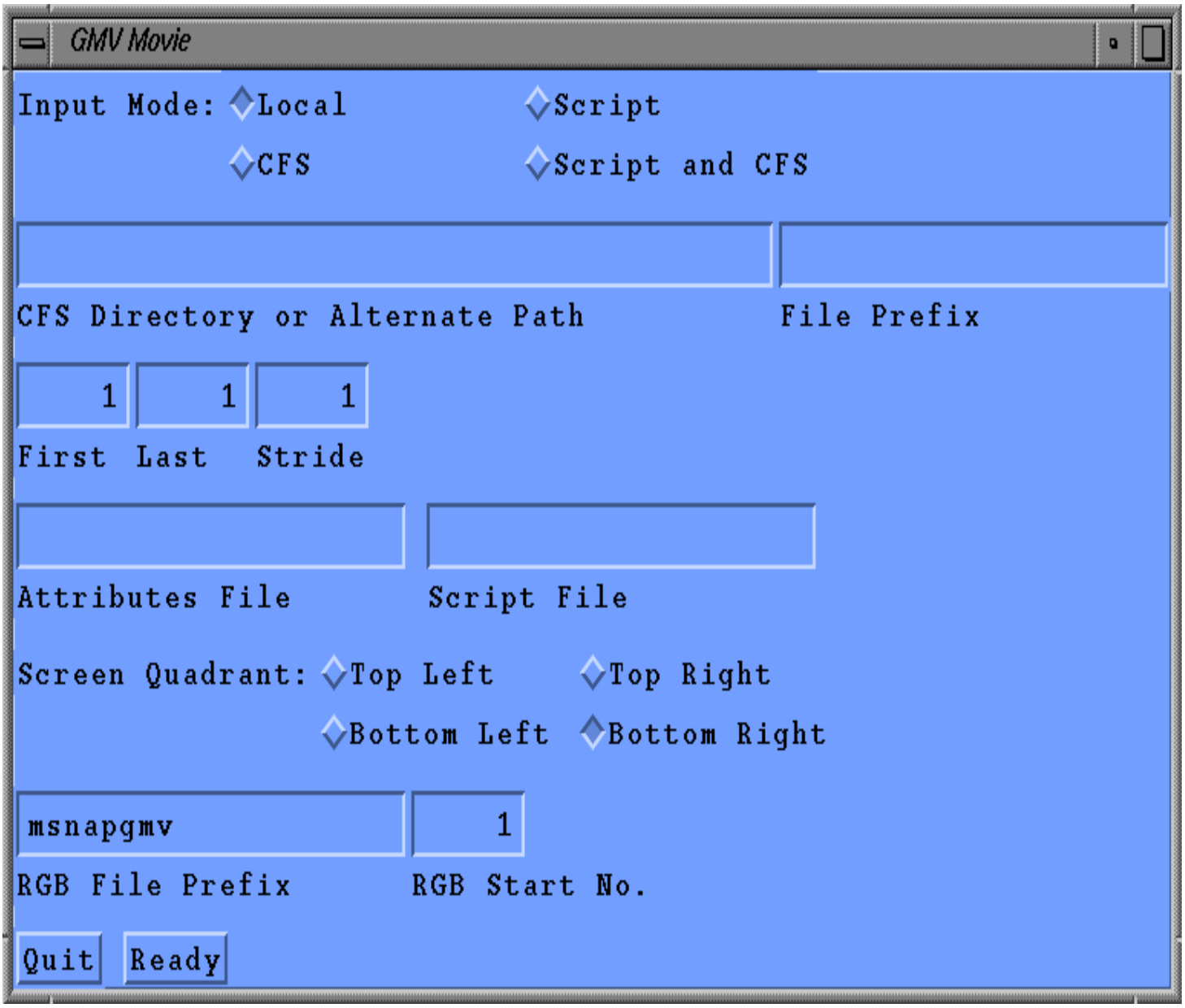


Figure 10−1. GMV Movie menu

The GMV Movie utility:

The GMVMOVIE utility can be used instead of a script to generate time−sequence movies, (see Fig 10−1). It works by first locating where the files are, then getting them one at a time and running gmv to make a snapshot image file of each frame. As GMVMOVIE finishes each frame, it removes the input file if the file was imported from CFS. If the input file was on a local disk, GMVMOVIE leaves it there.

After running GMVMOVIE, you may use the SGI utility "movie" to look at the snapshot frames interactively. The source for "movie" is provided in 4DGifts. The usual path name is ~4DGifts/iristools/imgtools.

Here are the steps for making a movie with GMVMOVIE:

1. Generate a series of gmv input files. These files must have the following naming scheme:

prefix + sequence,

where prefix is a string common to all the files (e.g.: gmva) and sequence is a 3−digit sequence number. For example, in the filename gmve023, the prefix is "gmve" and the sequence is 023. These files must reside on the Common File System (Los Alamos National Laboratory's mass storage system) or on a local disk.

2. Move one of these GMV input files to the workstation by using CFS and run gmv. Then adjust the image from this file to display relevant information for the attributes file. The attributes file will contain the viewing angles, magnification, visible materials, fields, cutplane parameters, and contour surfaces that will be applied to each frame of the movie. A helpful hint is to use a file late in the time sequence to set up attributes. An important point is that the range of values for any fields in this GMV file will subsequently apply to all frames of the movie. This makes the color−coding for cutplanes or bounding box fields consistent from frame to frame. Any field value in other files that is below the range in this file will be colored the minimum field color (dark blue); values above the range will be colored the maximum color (bright red).

When setup is complete, click on the "Put Attributes" button and specify a name for this attributes file. (Hint: use the suffix ".attr" to differentiate attributes files from other files.)

3. Start up GMVMOVIE and fill in the blanks on the menu, (see Fig. 10−1).

The Input Mode options are "Local", "CFS", "Script", "Script and CFS". "Local" means that the GMV input files are stored on the workstation, usually in the

current directory. "CFS" means the files are stored on CFS (LANL's Common File System mass storage system). "Script" means the files are stored locally, but the file prefix and the first, last, and stride values are read from the script file. This is useful in combining GMV input files with different file prefix names into one movie. "Script and CFS" is used when the files are stored on CFS and a script file determines the GMV input files to retrieve from CFS.

The box labeled "CFS Directory or Alternate Path" should contain the CFS path or Unix path (if different from the current directory). If the data is in the current directory, leave this box blank. Enter the prefix of the file names in the "File Prefix" box. For example, if the input files are stored on CFS node /my/mesa/node and the files are named gmvc001, ..., gmvc044, then the "CFS Directory or Alternate Path" box should contain:

/my/mesa/node

and the "File Prefix" box should contain:

gmvc

If you select "CFS" as the input mode, then GMVMOVIE will prompt you for a CFS password after you click on "Ready". If the CFS node has no password, press the Enter key. If a password is required, type it in after the prompt. It will not print on the screen.

Fill the "First", "Last", "Stride" boxes with the beginning, end, and interval of the GMV input files. For example, entering 1, 50, 5 with a prefix of gmvin1 will create movie frames for every fifth file from gmvin1001 through gmvin1050.

The "Attributes File" box should contain the attributes file name saved with the "Put Attributes" file option.

The "Script File" box is used when "Script" or "Script and CFS" input mode is selected. Enter the file name of the script file. For an example of a script that may be used here, please look under **Other Useful Information** below.

There are four options in the "Screen Quadrant" selection. Be sure that no other graphics window is running in the quadrant selected. The window that is drawn is 640 by 512 pixels; if the workstation does not have a 1280 by 1024 pixel resolution, then always select the upper left quadrant.

The "RGB File Prefix" box is used to enter the file name prefix of the snapshot files. The default name is "msnapgmv". This can be changed by the user. The "RGB Start No." contains the starting sequence number for the snapshot files. For example, if the "RGB File Prefix" contains rgbout and the "RGB Start No." is 50, then the snapshot file names will start with rbgout050. This is extremely helpful when a sequence has been interrupted and a restart is necessary.

Click on the "Ready" button to start the movie process.

4. After you click on "Ready", GMVMOVIE gets the first file and starts up GMV. If everything is correct, GMV will make an image in the selected quadrant of the screen, and then GMVMOVIE will save it in a file with the rgb file prefix and a three digit sequence number. You may do other work on the workstation while GMVMOVIE is running, but be careful not to obscure the selected quadrant of the screen. If this happens, whatever is put there might be captured in one of the snapshot image files.

5. The GMVMOVIE utility will output information in the executing window as it proceeds through its process to let you know what it's doing.

6. After GMVMOVIE finishes, you may look at the snapshot files it created by running the SGI utility "movie." To run movie, type:

movie (rgb file prefix)*

If you type:

movie msnapgmv* −z 2

it will fill the entire screen.

Other useful information

If you do not enter enough information, then when you select "Ready", the menu will remain and an error message appears. You must select one input mode and have the input file prefix and the attributes file name entered if input is local or from CFS. If the input mode is Script or Script and CFS the file prefix and the start, end, and stride data are read from the script file. The Script input mode can be used to combine GMV input files with different prefixes. For example, the following script data will process gmvfiles gmva001 to gmva005 then files gmvb001 to gmvb010:

```
gmva 1 5 1
gmvb 1 10 1
```

If you have GMV input files in your directory and they have the same names as files that GMVMOVIE will be getting from CFS, the existing input files will be overwritten.

You can run up to four gmvmovies at a time by selecting different

quadrants for each movie, as long as the workstation has a 1280 x 1024 pixel resolution. Just make sure that the quadrants do not overlap and there are no other graphics images or menus running in the quadrants.

If you cannot get GMVMOVIE to stop, go to another window and type

```
ps −ef | grep gmvmovie
```

You will see something like this:

```
jxf  5503  4991 80 11:35:36 ttyq9     0:45 gmvmovie
jxf  5507  5471  1 11:36:28 console   0:00 grep gmvmovie
```

Locate the process number for GMVMOVIE (in this case, 5503) and then kill it by typing

kill (process number), e.g.: kill −9 5503.

If you are running the OpenGL version of GMV on an SGI, the window manager will require you to place the window on the screen, contrary to the −w option. To prevent this interactive placement problem, add the following lines to your .Xdefaults file, then log in to activate the resources.

```
4DWm*interacitvePlacement:false
4DWm*GMV*clientDecoration:none
```

GMVBATCH

GMVBATCH − Off Screen Rendering:

GMVBATCH is a command line version of GMV that allows off screen rendering of an image using Mesa's OSMesa off screen image buffers. A SGI RGB file is then generated. GMVBATCH is a better way to generate movies than regular GMV because it does not need an OpenGL window to draw into, thus a script can be written to use GMVBATCH and run in the background.

GMVBATCH command line options:

GMVBATCH has several command line options which are used to create time sequence movies of mesh data, noninteractively. The following are the available GMVBATCH command line options:

window size: ***−w xloc yloc width height***

xloc, yloc: X and Y coordinates of lower left corner of "window" where GMVBATCH will generate the image.
width, height: the width and height in pixels where GMVBATCH will generate the image.
Default: 0 0 900 700 Note, xloc and yloc are not used by GMVBATCH.

input file name: ***−i filename***

Specifies an input file for GMVBATCH.

attributes file name: ***−a attribute_filename***

Specifies an attributes file for GMVBATCH to use to draw the image.

snapshot file: ***−s snapshot_filename***

Saves the image in SGI RGB format to the file specified in −s. If −s is not entered, the file name "AzsnapgmvAz" is saved as the default.

Here is an example script that uses GMVBATCH to create a series of snapshots for use in movie making:

```
#
set verbose
set k=(1−21)
foreach i (k)
gmvBatch −a iso.attr −w 639 0 640 512 −i gmvout3.$i −s ksbmiso.$i.rgb
end
```

Helpful Hints

1) Nodes are more visible with a black background. To change the background color, select "Background Color" under "Color Edit" . All three colors sliders (red, green, and blue) must be all the way to the left for a black background.

2) Shaded objects such as polygons and isosurfaces look better with a white background. All three color controls must be to the right for a white background.

3) When selecting polygons, turn off the "Shaded" and "Edges" buttons. First select everything you need to view, and *then* shade the polygons. This way, GMV only has to update the screen once, instead of every time you change your selection. This is especially helpful for large problems.

4) Hint number three also applies for cells. Turn everything off and select what you need first. Then turn on faces, or edges as needed.

5) Workstations without any graphics acceleration or ones running the Mesa (OpenGL to X) version are especially slow running GMV. Therefore, it is to your advantage not to overwork the machine and only force GMV to do something when it is necessary. The mouse controls are not very useful when running the slow version of GMV. It is more efficient to use the slider bars above the main viewer to manipulate the object. This will be more accurate and save time. Starting GMV with a smaller window will also speed up the interactive drawing process. For example, you could invoke GMV with the **−w** option like this:

gmv −w 0 0 500 400

which will start GMV with a 500x400 pixel main viewer instead of the usual 900x700 pixel main viewer. Setting the interactivity slider bar to its highest setting is helpful. Thus, when you manipulate the object in the main viewer, less data will display until manipulations are complete. To guide you during this process, turn on the bounding box as a reference tool. It will not disappear during the interactive drawing process as the data does.

6) There are many image conversion utilities available that can convert SGI RGB to other image formats. One is called ***imtools*** and is available at ***ftp.sdsc.edu*** (the San Diego Super Computing Center). The compressed tape archive of these utilities is about 11 megabytes. The relevant tool included in this package is called ***imconv***. This tool can convert files into practically any format from any format.

Another helpful tool for image conversion is xv. The URL for this tool is: ***http://www.sun.com/sunsogt/catlink/xv/xv.html***.

7) Material isosurfaces: GMV will generate material isosurfaces using node values. There must be a change in material values in a cell before isosurface elements can be generated for the cell. It may be better for you to generate your own material surfaces and enter them in the ***polygons*** list.

8) Null material number: Nodes or cells with a 0 (zero) material number are not drawn and are not included when calculating field minimums and maximums. A "zero" material number can be useful if your simulation deletes nodes but keeps a continuous numbering sequence. You can keep deleted nodes so you do not have to renumber cell vertices; just enter a 0 (zero) material number for those nodes, and GMV will ignore them unless requested.

9) If polygons are the only necessary elements of your simulation, you must still enter data for nodes, cells and materials. There may be zero cells, but there must be at least one node. Place this node somewhere within the polygon boundaries. In the materials list, the single node must be assigned a material number. An example GMV input file follows:

gmvinput **ascii**
nodes 1
0
0
0
cells 0
material 3 1
mat1
mat2
mat3
1
polygons
{ polygon data }
endpoly

endgmv

10) Menu fonts: You can use standard X resource files to change the font or font size of the menus. Using a smaller font is useful when you start GMV with a small drawing window using the −w option. An example of the resource line for a 14 point font follows:

GMV*fontList: −adobe−courier−bold−r−*−*−14−*

The following example line will cause the initial File Selection menu to have a different pattern.

GMV*pattern: *gmv*

For example, you can place the lines above in a file called Resources, then set the XENVIRONMENT variable to point to the Resources file. Here is a c−shell example:

setenv XENVIRONMENT Resources

11) Faster remote OpenGL rendering: Be sure to turn on the "Use Display List" option under the "Controls−3" menu to get faster interactivity with rotation, pan, and zoom features when running GMV over the network. Note, the Display List option is on by default.

12) To save memory with version 2.2 and above, turn off the Display List option in the gmvrc file before starting GMV..

Acknowledgments

Special thanks are in order for the following people:

John D. Fowler Jr. (CIC Division):
for creating the initial port to OpenGL.

Harold Trease,
Manjit Sahota,
John McGhee,
Mabel Grey,
Paul Maudlin:
for ideas and initial tests.

Jeff Hinrichs,
Rebecca Fresquez,
Kevin Bolling:
for their contributions in creating this manual.

Glen Hansen:
for assisting with the maintenance of GMV and this manual.

Los Alamos

NATIONAL LABORATORY

Los Alamos, New Mexico 87545